

T U T O R I A L S

Version 8.0

SMS 8.0

Copyright © 2002 Brigham Young University – Environmental Modeling Research Laboratory March 15, 2002.

All Rights Reserved

Unauthorized duplication of the *SMS* software or user's manual is strictly prohibited.

THE BRIGHAM YOUNG UNIVERSITY ENVIRONMENTAL MODELING RESEARCH LABORATORY MAKES NO WARRANTIES EITHER EXPRESS OR IMPLIED REGARDING THE PROGRAM *SMS* AND ITS FITNESS FOR ANY PARTICULAR PURPOSE OR THE VALIDITY OF THE INFORMATION CONTAINED IN THIS USER'S MANUAL

The software *SMS* is a product of the Environmental Modeling Research Laboratory of Brigham Young University

www.emrl.byu.edu

Last Revision: March 15, 2002

TABLE OF CONTENTS

1	INTRODUCTION	1-1
1.1	SMS HELP	1-2
1.2	SUGGESTED ORDER OF CO MPLETION	1-2
1.3	MODULE	1-3
1.4	DEMO VS. WORKING MODES	1-4
2	OVERVIEW OF SMS.....	2-1
2.1	INTRODUCTION	2-1
2.2	GETTING STARTED	2-1
2.3	THE SMS SCREEN	2-2
2.3.1	THE MAIN GRAPHICS WINDOW	2-2
2.3.2	THE TOOLBOX	2-2
2.3.3	THE EDIT WINDOW	2-3
2.3.4	THE MENU BAR.....	2-3
2.4	USING A BACKGROUND IMAGE	2-4
2.4.1	IMPORTING THE IMAGE.....	2-4
2.4.2	REGISTERING THE IMAGE	2-4
2.4.3	RESAMPLING THE IMAGE.....	2-4
2.5	USING FEATURE OBJECTS	2-5
2.6	CREATING FEATURE ARCS	2-6
2.7	MANIPULATING COVERAGES	2-8
2.8	REDISTRIBUTING VERTICES	2-9
2.9	DEFINING POLYGONS	2-10
2.10	ASSIGNING MESHING PARAMETERS	2-10
2.10.1	CREATING A REFINE POINT FOR ADAPTIVE TESSELLATION	2-10
2.10.2	DEFINING A COONS PATCH	2-11
2.10.3	REMOVING DRAWING OBJECTS.....	2-13
2.11	APPLYING BOUNDARY CONDITIONS	2-14
2.11.1	DEFINING ARC GROUPS	2-14
2.11.2	ASSIGNING THE BOUNDARY CONDITIONS	2-15
2.12	ASSIGNING MATERIALS TO POLYGONS.....	2-16
2.12.1	DISPLAYING MATERIAL TYPES	2-16
2.13	CONVERTING FEATURE OBJECTS TO A MESH	2-17
2.14	EDITING THE FEATURE OBJECT MESH.....	2-18
2.15	INTERPOLATING TO THE MESH	2-18
2.16	RENUMBERING THE MESH.....	2-19
2.17	SAVING A PROJECT FILE.....	2-20
2.18	CONCLUSION	2-20
3	MESH EDITING	3-1
3.1	IMPORTING TOPOGRAPHIC DATA	3-1
3.2	TRIANGULATING THE NODES	3-2
3.3	DELETING OUTER ELEMENTS.....	3-3
3.4	DELETING THIN TRIANGLES.....	3-3
3.5	MERGING TRIANGLES	3-4
3.6	EDITING INDIVIDUAL ELEMENTS.....	3-5
3.6.1	USING THE SPLIT / MERGE TOOL	3-6
3.6.2	USING THE SWAP EDGE TOOL	3-7
3.7	SMOOTHING THE BOUNDARY	3-9

3.8	RENUMBERING THE MESH.....	3-11
3.9	CHANGING THE CONTOUR OPTIONS	3-11
3.10	CHECKING THE MESH QUALITY	3-12
3.11	REFINING ELEMENTS.....	3-15
3.11.1	INSERTING BREAKLINES	3-16
3.11.2	USING THE REFINEMENT COMMAND.....	3-18
3.12	FINISHING THE MESH	3-19
3.13	SAVING THE MESH	3-19
3.14	CONCLUSION.....	3-19
4	BASIC RMA2 ANALYSIS	4-1
4.1	INTRODUCTION	4-1
4.2	DEFINING MATERIAL PROPERTIES	4-2
4.3	CHECKING THE MODEL.....	4-3
4.4	SAVING THE SIMULATION.....	4-3
4.5	USING GFGEN	4-3
4.6	USING RMA2.....	4-4
4.7	CONCLUSION.....	4-4
5	BASIC FESWMS ANALYSIS.....	5-1
5.1	INTRODUCTION	5-1
5.2	CONVERTING ELEMENTS	5-2
5.3	DEFINING MATERIAL PROPERTIES	5-2
5.4	SETTING MODEL CONTROLS	5-3
5.5	MODEL CHECK.....	5-4
5.6	SAVING THE SIMULATION.....	5-4
5.7	USING FLO2DH	5-5
5.8	CONCLUSION.....	5-5
6	2D POST PROCESSING	6-1
6.1	INTRODUCTION	6-1
6.2	DATA SETS.....	6-1
6.3	USING THE DATA BROWSER	6-2
6.4	CREATING NEW DATA SETS WITH THE DATA CALCULATOR.....	6-3
6.5	CREATING ANIMATIONS.....	6-3
6.5.1	CREATING A FILM LOOP ANIMATION	6-4
6.5.2	CREATING A FLOW TRACE ANIMATION	6-6
6.6	DROGUE PLOT ANIMATION	6-7
6.7	2D PLOTS	6-8
6.8	CONCLUSION.....	6-8
7	ADVANCED RMA2 ANALYSIS.....	7-1
7.1	INTRODUCTION	7-1
7.2	DEFINING MATERIAL PROPERTIES	7-2
7.3	CREATING NODESTRINGS.....	7-2
7.4	DEFINING A DYNAMIC SIMULATION.....	7-3
7.4.1	DEFINING TIME PARAMETERS	7-3
7.4.2	DEFINING HOTSTART OUTPUT	7-4
7.4.3	DEFINING ITERATION CONTROLS	7-4
7.4.4	DEFINING WETTING/DRYING PARAMETERS	7-4
7.5	DEFINING TRANSIENT BOUNDARY CONDITIONS	7-5
7.5.1	DEFINING FLOW BOUNDARY CONDITIONS	7-5
7.5.2	DEFINING HEAD BOUNDARY CONDITIONS	7-5
7.6	RUNNING THE MODEL CHECKER	7-6
7.7	SAVING THE SIMULATION.....	7-6
7.8	USING REVISION RECORDS	7-7
7.9	RUNNING THE INITIAL SIMULATION	7-7

7.9.1	RUNNING GFGEN.....	7-7
7.9.2	RUNNING RMA2.....	7-8
7.10	DEFINING A STEADY STATE SIMULATION	7-8
7.11	DEFINING CONSTANT BOUNDARY CONDITIONS.....	7-9
7.12	SAVING THE NEW SIMULATION	7-9
7.13	RUNNING THE FINAL SIMULATION.....	7-9
7.14	CONCLUSION.....	7-10
8	ADVANCED FESWMS ANALYSIS	8-1
8.1	INTRODUCTION	8-1
8.2	DEFINING MATERIAL PROPERTIES	8-2
8.3	CREATING THE HOTSTART FILE.....	8-3
8.3.1	ASSIGNING BOUNDARY CONDITIONS	8-3
8.3.2	CREATING WEIRS	8-4
8.3.3	SAVING THE DATA	8-6
8.3.4	USING FLO2DH	8-7
8.4	REWORKING THE SOLUTION.....	8-7
8.4.1	CHANGING THE BOUNDARY CONDITIONS	8-8
8.4.2	EDITING THE WEIR DATA.....	8-8
8.4.3	USING THE HOT START FILE	8-8
8.4.4	COMPUTING A NEW SOLUTION FILE USING A HOTSTART FILE.....	8-9
8.5	CONCLUSION.....	8-9
9	SED2D-WES ANALYSIS	9-1
9.1	TABS DATA FLOW	9-1
9.1.1	GFGEN	9-2
9.1.2	RMA2	9-2
9.1.3	SED2D-WES	9-3
9.2	RMA2 COLDSTART SIMULATION.....	9-3
9.3	RMA2 RATING CURVE SIMULATION	9-6
9.4	INITIAL SED2D-WES SIMULATION	9-8
9.5	RMA2/SED2D HYDROGRAPH ANALYSIS	9-10
9.5.1	UPDATES FOR RMA2.....	9-10
9.5.2	UPDATES FOR SED2D.....	9-11
9.5.3	SAVING THE PROJECT	9-11
9.5.4	RUNNING THE SIMULATION	9-12
9.6	CONCLUSION.....	9-12
10	RMA4 ANALYSIS.....	10-1
10.1	INTRODUCTION	10-1
10.2	CASE 1	10-1
10.2.1	RMA4 MODEL CONTROL	10-2
10.2.2	BOUNDARY CONDITIONS	10-3
10.2.3	MATERIAL PROPERTIES	10-3
10.2.4	RUN RMA4.....	10-4
10.2.5	FILM LOOP	10-5
10.3	CASE 2	10-6
10.3.1	RMA4 MODEL CONTROL	10-6
10.3.2	BOUNDARY CONDITIONS	10-7
10.3.3	MATERIAL PROPERTIES	10-7
10.3.4	RUN RMA4.....	10-7
10.3.5	FILM LOOP	10-8
10.4	CASE 3	10-8
10.4.1	RMA4 MODEL CONTROL	10-8
10.4.2	BOUNDARY CONDITIONS	10-9
10.4.3	MATERIAL PROPERTIES	10-9
10.4.4	RUN RMA4.....	10-9
10.4.5	FILM LOOP	10-10
10.5	OTHER CHANGES	10-10

10.6	CONCLUSION	10-10
11	HIVEL ANALYSIS.....	11-1
11.1	INTRODUCTION	11-1
11.2	CREATING MATERIALS	11-2
11.3	CREATING NODESTRINGS	11-2
11.4	DEFINING BOUNDARY CONDITIONS	11-3
11.4.1	GENERAL PARAMETERS	11-3
11.4.2	DEFINING STEADY STATE FLOW AND HEAD.....	11-3
11.4.3	CREATING THE HOTSTART FILE	11-4
11.5	SAVING THE SIMULATION.....	11-5
11.6	USING HIVEL2D	11-5
11.7	CONCLUSION.....	11-6
12	CGWAVE ANALYSIS	12-1
12.1	INTRODUCTION	12-1
12.2	CREATING A WAVELENGTH FUNCTION	12-1
12.3	CREATING A SIZE FUNCTION.....	12-2
12.4	DEFINING THE DOMAIN	12-3
12.4.1	CREATING THE COASTLINE	12-3
12.4.2	CREATING THE DOMAIN	12-4
12.5	CREATING THE FINITE ELEMENT MESH.....	12-4
12.5.1	SETTING UP THE POLYGON.....	12-5
12.5.2	GENERATING THE ELEMENTS	12-5
12.6	MODEL CONTROL	12-6
12.7	RENUMBERING.....	12-7
12.8	SAVING THE CGWAVE DATA.....	12-7
12.9	RUNNING CGWAVE	12-7
12.10	CONCLUSION	12-8
13	ADCIRC ANALYSIS.....	13-1
13.1	INTRODUCTION	13-1
13.2	READING IN A COASTLINE FILE.....	13-1
13.2.1	DEFINING THE DOMAIN	13-2
13.2.2	ASSIGNING BOUNDARY TYPES.....	13-2
13.3	EDITING THE COASTLINE FILE	13-3
13.3.1	COORDINATE CONVERSIONS	13-3
13.4	READING IN A SHOALS FILE.....	13-4
13.5	SHALLOW WAVELENGTH FUNCTIONS	13-5
13.6	CREATING SIZE FUNCTIONS	13-6
13.6.1	FINDING THE CENTRAL POINT FOR THE MESH	13-6
13.6.2	DISTANCE FUNCTION	13-8
13.6.3	INITIAL SIZE FUNCTION	13-8
13.6.4	SCALE FUNCTION	13-8
13.6.5	FINAL SIZE FUNCTION	13-9
13.7	CREATING POLYGONS	13-9
13.7.1	BUILDING POLYGONS	13-10
13.7.2	POLYGON ATTRIBUTES	13-10
13.7.3	ASSIGNING THE MESHING TYPE.....	13-10
13.7.4	ASSIGNING THE BATHYMETRY TYPE.....	13-10
13.7.5	ASSIGNING THE POLYGON TYPE	13-11
13.8	CREATING THE MESH.....	13-11
13.8.1	MESH DISPLAY OPTIONS	13-11
13.8.2	MINIMIZING MESH BANDWIDTH	13-12
13.9	BUILDING THE ADCIRC CONTROL FILE.....	13-12
13.9.1	CONVERTING BACK TO LAT/LON	13-13
13.9.2	MAIN MODEL CONTROL SCREEN	13-13
13.9.3	TIME CONTROL	13-14

13.9.4	TIDAL FORCES	13-14
13.9.5	SAVING THE MESH AND CONTROL FILES	13-15
13.10	RUNNING <i>ADCIRC</i>	13-15
13.11	IMPORTING <i>ADCIRC</i> GLOBAL OUTPUT FILES	13-16
13.12	VIEWING <i>ADCIRC</i> OUTPUT.....	13-16
13.12.1	SCALAR DATASET OPTIONS.....	13-17
13.12.2	VECTOR DATASET OPTIONS.....	13-18
13.13	FILM LOOP VISUALIZATION.....	13-20
13.14	CONCLUSION	13-20
14	STWAVE ANALYSIS	14-1
14.1	INTRODUCTION	14-1
14.2	CONVERTING <i>ADCIRC</i> TO SCATTER	14-1
14.2.1	READING IN THE <i>ADCIRC</i> FILES.....	14-1
14.2.2	CONVERTING TO SCATTER	14-2
14.3	CREATING THE CARTESIAN GRID	14-3
14.3.1	CREATING THE CARTESIAN GRID FRAME	14-3
14.3.2	CREATING THE LAND AND OCEAN POLYGONS.....	14-4
14.3.3	MAPPING TO THE GRID	14-5
14.4	EDITING THE GRID AND RUNNING STWAVE	14-6
14.4.1	GENERATING SPECULAR ENERGY DISTRIBUTION	14-6
14.4.2	MODEL CONTROL.....	14-7
14.4.3	SELECTING MONITORING STATIONS.....	14-7
14.4.4	SAVING THE SIMULATION	14-7
14.4.5	RUNNING STWAVE	14-8
14.5	POST PROCESSING.....	14-8
14.5.1	VISUALIZING THE STWAVE SOLUTION.....	14-8
14.5.2	VISUALIZING BATHYMETRY	14-8
14.5.3	VISUALIZING THE DIRECTION FIELD	14-9
14.5.4	VISUALIZING THE WAVE HEIGHT.....	14-9
14.5.5	VISUALIZING THE OBSERVATION (MONITORING CELLS) SPECTRA	14-9
14.6	CONCLUSION	14-9
15	HEC-RAS ANALYSIS.....	15-1
15.1	INTRODUCTION	15-1
15.2	PREPARING THE CONCEPTUAL MODEL	15-1
15.2.1	CREATING THE COVERAGES.....	15-2
15.2.2	CREATING CENTERLINE AND BANK ARCS.....	15-3
15.2.3	NAMING THE CENTERLINE ARCS.....	15-4
15.2.4	CREATING LAND USE COVERAGE	15-5
15.2.5	CREATING THE CROSS-SECTIONS.....	15-6
15.2.6	EXTRACTING CROSS-SECTIONS.....	15-7
15.3	CREATING THE NETWORK SCHEMATIC	15-8
15.3.1	CREATING THE GEOMETRY IMPORT FILE	15-10
15.4	USING <i>HECRAS</i>	15-11
15.5	POST PROCESSING.....	15-12
15.5.1	PROFILE PLOTS.....	15-13
15.5.2	CROSS SECTION PLOTS	15-13
15.5.3	POST PROCESSING EXPERIMENTATION	15-14
15.6	CONCLUSION	15-14
16	BRI-STARS ANALYSIS	16-1
16.1	INTRODUCTION	16-1
16.2	READING IN THE IMAGE.....	16-1
16.3	CREATING A CONCEPTUAL MODEL.....	16-2
16.3.1	DEFINING A CENTERLINE.....	16-2
16.3.2	CREATING BANK ARCS.....	16-3
16.3.3	CREATING CROSS SECTIONS	16-4
16.3.4	CREATING AN AREA PROPERTY COVERAGE	16-5

16.3.5	ASSIGNING MATERIAL TYPES	16-6
16.3.6	IMPORTING TOPOGRAPHIC DATA	16-7
16.3.7	EXTRACTING THE CROSS SECTIONS	16-8
16.3.8	ASSIGNING REACH ATTRIBUTES	16-9
16.3.9	ASSIGNING CROSS SECTION ATTRIBUTES.....	16-9
16.3.10	SAVING THE DATA.....	16-10
16.3.11	CREATING THE SCHEMATIC	16-10
16.4	1D RIVER HYDRAULIC MODULE	16-11
16.4.1	ASSIGNING MATERIAL VALUES	16-11
16.4.2	DEFINING THE CONTROL PARAMETERS	16-12
16.4.3	SAVING THE SIMULATION	16-14
16.5	RUNNING BRI-STARS	16-14
16.6	POST PROCESSING.....	16-14
16.6.1	OPENING THE OUTPUT FILES.....	16-14
16.6.2	CREATING PLOTS.....	16-15
16.7	CONCLUSION.....	16-16
17	OBSERVATION COVERAGE	17-1
17.1	INTRODUCTION	17-1
17.2	OPENING THE DATA	17-1
17.3	VIEWING SOLUTION DATA	17-2
17.4	CREATING AN OBSERVATION COVERAGE	17-2
17.5	THE OBSERVATION COVERAGE.....	17-3
17.5.1	CREATING A MEASUREMENT	17-3
17.6	CREATING AN OBSERVATION POINT	17-4
17.6.1	USING THE CALIBRATION TARGET	17-6
17.6.2	MULTIPLE MEASUREMENTS.....	17-7
17.7	READING A SET OF OBSERVATION POINTS	17-8
17.8	GENERATING ERROR PLOTS	17-9
17.8.1	PLOT DATA	17-10
17.8.2	USING THE COMPUTED VS. OBSERVED DATA PLOT.....	17-10
17.8.3	USING THE ERROR SUMMARY PLOT	17-11
17.9	CALIBRATING THE MODEL	17-11
17.9.1	EDITING THE MATERIAL PROPERTIES	17-11
17.9.2	COMPUTING A NEW SOLUTION	17-12
17.9.3	READING THE NEW SOLUTION.....	17-12
17.9.4	FINE-TUNING THE MODEL.....	17-12
17.10	USING THE ERROR VS. SIMULATION PLOT	17-13
17.11	GENERATING OBSERVATION PROFILE PLOTS	17-15
17.12	GENERATING TIME SERIES PLOTS.....	17-17
17.13	CONCLUSION	17-18
18	SENSITIVITY ANALYSIS	18-1
18.1	INTRODUCTION	18-1
18.2	SIMPLE CHANNEL WITH SINGLE MATERIAL	18-1
18.2.1	OPEN THE SIMULATION	18-1
18.2.2	RUNNING THE MODEL.....	18-2
18.2.3	IMPORTING SOLUTIONS.....	18-2
18.2.4	CREATING PROFILE PLOTS	18-2
18.2.5	VARYING MANNING'S ROUGHNESS	18-3
18.2.6	UPDATING THE PLOT.....	18-4
18.2.7	CHANGES IN EDDY VISCOSITY	18-4
18.3	CONSTRAINED FLUME WITH SINGLE MATERIAL.....	18-6
18.3.1	OPEN THE SIMULATION	18-6
18.3.2	VARYING MANNING'S COEFFICIENT.....	18-6
18.3.3	VARYING EDDY VISCOSITIES	18-7
18.4	SIMPLE CHANNEL WITH TWO MATERIALS.....	18-8
18.5	CONCLUSION.....	18-10

Introduction

This document contains tutorials for the Surface-water Modeling System (*SMS*) version 8.0. Each tutorial is meant to provide training on a specific component of *SMS*. It is strongly suggested that you complete the tutorials before using *SMS* on a routine basis. For addition training, contact your *SMS* distributor.

SMS is a pre- and post-processor for surface water modeling and analysis. It includes two- and three-dimensional finite element and finite difference models, and one-dimensional step backwater modeling tools. Interfaces specifically designed to facilitate the utilization of several numerical models comprise the modules of *SMS*. Supported models include the USACE-WES supported *TABS-MD* (*GFGEN*, *RMA2*, *RMA4*, *RMA10*, *SED2D-WES*), *HIVEL2D*, *ADCIRC*, *CGWAVE*, *STWAVE*, and *M2D*. Comprehensive interfaces have also been developed for facilitating the use of the FHWA commissioned analysis packages *Flo2dh* and *WSPRO*.

Each numerical model is designed to address a specific class of problem. Some calculate hydrodynamic data such as water surface elevations and flow velocities. Others compute wave mechanics such as wave height and direction. Still others track contaminant migration or suspended sediment concentrations. Some of the models support both steady-state and dynamic analyses, while others support only steady-state analysis. Some support supercritical flow, while others support only subcritical.

The finite element mesh, finite difference grid, or cross section entities, along with associated boundary conditions necessary for analysis, are created within *SMS* and then saved to model-specific files. These files are used as input to the hydrodynamic, wave mechanic, contaminant migration, and sediment transport analysis engines. The numerical models create solution files that contain the water surface elevations, flow

velocities, contaminant concentrations, sediment concentrations or other functional data at each node, cell, or section. *SMS* reads this data to create plots and animations.

SMS can also be used as a pre- and post-processor for other finite element or finite difference programs as long as the programs can read and write files in a supported format. To facilitate this, a *generic* interface is available to define parameters for a proprietary model. *SMS* is well suited for the construction of large, complex meshes (up to hundreds of thousands of elements) of arbitrary shape.

Please note that in these tutorials, reference to a menu item will be as follows: *Menu | Menu-Item*. For example: *File | Quit* indicates to select the *Quit* item from the *File* menu.

1.1 SMS Help ---

Accompanying *SMS* is the *SMS online Help*, which fully describes the available options in each dialog box. The help can be accessed through the *Help* menu inside of *SMS* or from the *Help* button on each dialog box. In addition, help files are available for some of the numerical models supported by *SMS*.

1.2 Suggested Order of Completion ---

Most of these lessons are developed for two dimensional finite element meshes and finite difference grids. If you want to use the *SMS* for its *River Hydraulic Module* and *HECRAS* interface (including *WSPRO*), you may want to examine the first tutorial (Lesson 2) and then skip to Lessons 15 and 16.

For other users of *SMS*, we recommend that you start with Lessons 2 and 3, which describe the basic tools available in *SMS* for generating finite element meshes. From there, a variety of lessons could be completed, depending on the numerical model you wish to use. Lessons 4, 7, 9 and 10 use the TABS models. Lessons 5 and 8 use the Flo2DH model. Finally, lessons 12, 13 and 14 use the wave models CGWAVE, ADCIRC and STWAVE, respectively.

Lessons 6, 17 and 18 concentrate on using the post-processing capabilities of *SMS*. One of these uses the *RMA2* results while another uses the *Flo2dh* results. However, the techniques presented are identical for post-processing all numerical models. The output files required to complete each lesson have all been provided, so it is possible to complete any lesson, whether or not *SMS* has been licensed.

1.3 Modules

The *SMS* interface is divided into six modules. A module is provided for each of the basic data types supported by *SMS*. As you switch from one module to another, the tools in the *Toolbox* and the menus change. This division allows for focus on small portions of the interface at a time. The *Mesh* and *River* modules have interfaces to certain numerical analysis models. For these modules, only one numerical model is available at a time.

1.4 Demo Vs. Working Modes

Some users do not require all modules or model interfaces provided in *SMS*, which can be licensed individually. The icons for unlicensed modules and the menus for unlicensed model interfaces cannot be accessed. It is possible, however, to access all modules and model interfaces in *SMS* by running in *Demo Mode*.

If you have not licensed any part of *SMS*, it will automatically run in demo mode. On the other hand, if you do have a license for part of *SMS* and would like to experiment with an unlicensed module, you can tell *SMS* to run in demo mode. To do this:

1. Select *File / Demo mode*.
2. Select *Yes* to the prompt, *Are you sure you want to delete everything?*

When you do this, a check mark appears next to this menu command. Demo mode in *SMS* allows access to all functions except the *Save* and *Print* commands. To get back to normal operation mode, select this menu item a second time – you will once again be required to delete all data. Note that if you have no registered modules, you cannot leave demo mode.

Overview

2.1 Introduction

This tutorial describes the major components of the *SMS* interface and gives a brief introduction to the different *SMS* modules. It is suggested that this tutorial be completed before any other tutorial. All files for this tutorial are in the *tutorial\tut2* directory.

2.2 Getting Started

Before beginning this tutorial you should have installed *SMS* on your computer. If you have not yet installed *SMS*, please do so before continuing. Each chapter of this tutorial document demonstrates the use of a specific component of *SMS*. If you have not purchased all modules of *SMS*, or if you are evaluating the software, you should run *SMS* in *Demo Mode* to complete this tutorial (see section 1.4). When using *Demo Mode*, you will not be able to save files. For this reason, all files that you are asked to save have been included in the *output* subdirectory under the *tutorial\tut2* directory. When you are asked to save a file, you should instead open the file from this *output* directory. To start *SMS*, *do the following*:

- Open the *Start* menu, scroll to *Programs*, and then to *SMS* and click on *SMS 8.0*.

2.3 The SMS Screen

The *SMS* screen is divided into five main sections: the *Main Graphics Window*, the *Toolbox*, the *Edit Window*, the *Menu Bar* and the *Status Bars*, as shown in Figure 2-1. In addition, plot windows can be opened to display 2D plots of various data.

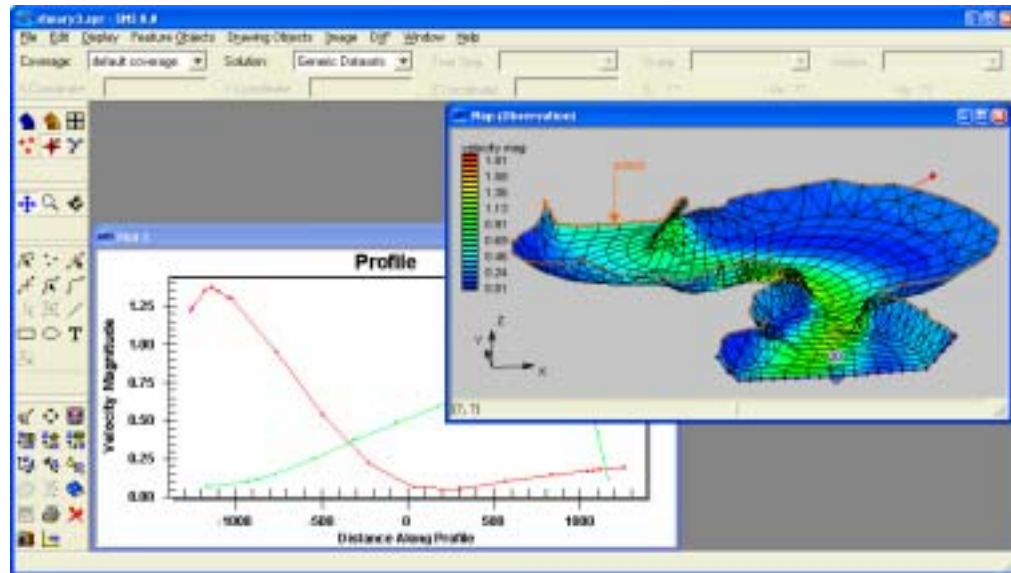


Figure 2-1. The SMS screen.

2.3.1 The Main Graphics Window

The *Main Graphics Window* is the biggest part of the *SMS* screen. Most of the data manipulation is done in this window. You will use it with each tutorial chapter.

2.3.2 The Toolbox

The *Toolbox* is broken into the following four docking toolbars:

- *Modules*. There are currently six *SMS Modules*. The functions of these are described in the *SMS online Help*.



- *Static Tools*. This contains a set of tools that do not change for different modules. These tools are used for manipulating the display.



- *Dynamic Tools*. These tools change according to the selected module. These tools are used for creating and editing entities specific to the module.

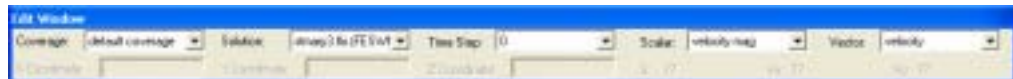


- *Macros*. These are shortcuts for frequently used menu commands.



2.3.3 The Edit Window

The *Edit Window* is used to show and/or change the coordinates of selected entities as well as manipulate the active coverage, solution, scalar and vector data sets and the active solution time step.



2.3.4 The Menu Bar

The *Menu Bar* contains commands that are available for data manipulation. The menus shown in the *Menu Bar* depend on the active module and numerical model.

2.3.5 The Status Bars

There are two status bars: one at the bottom of the SMS application window and a second attached to the *Main Graphics Window*. The status bar attached to the main application window shows help messages when the mouse hovers over a tool or an item in a dialog box. A times, it also may display a message in red text to prompt for specific actions, such as that shown in the figure below.



The second status bar, attached to the *Main Graphics Window*, is split into two separate panes. The left shows the mouse coordinates when the model is in plan view. The right pane shows information for selected entities.



2.4 Using a Background Image

A good way to visualize the model is to import a digital image of the site. For this example, an image was created by scanning a portion of a USGS quadrangle map and saving the scanned image as a *TIFF* file. SMS can open and display TIFF and JPEG images. Once the image is inside *SMS*, it is displayed in plan view behind all other data.

2.4.1 Opening The Image

To open the TIFF image:

1. Select *File /Open*.
2. Open the file *stmary.tif* from the *tutorial\tut2* directory. A preview image will be shown in the *Register Image* dialog. (Do not click the *OK* button yet.)

2.4.2 Registering The Image




All TIFF images must be registered after being opened, except for *geo-referenced* TIFF images. If the *Register Image* dialog does not appear when the image is opened, then the image is geo-referenced and does not need to be registered. To register a TIFF image, three points on the image, called *registration points*, are given real world coordinates. When the image is drawn, it is skewed, rotated, and stretched according to the registration points. The registration points can be defined manually or can be read from a *TIFF world file*. For this example, a TIFF world file will define the registration points. To open the TIFF world file:

1. In the *Register Image* dialog click the *Import World File* button.
2. Open the file *stmary.tfw*. The three register points (red cross-hairs) will be assigned real world coordinates.
3. Click the *OK* button to close the *Register Image* dialog.

A progress bar appears in the *Status Bar* along with a message that the image is being resampled. After a few moments, the image will be drawn in the *Graphics Window*. The image will always be drawn under all other objects.

2.4.3 Resampling The Image

When zooming in or out, the image can become poorly sampled. It can be cleared up by resampling. To demonstrate this:

1. Choose the *Zoom*  tool from the *Toolbox*.
2. Drag a box around the main river portion, as shown in Figure 2-2. The image is somewhat blurry because of the old image sampling.
3. Select the *Resample*  macro button. This will resample the image to display only the zoomed part and can enhance the image resolution.
4. Alternately, an image can be resampled by changing to the *Map*  module and selecting *Image / Resample*.

Note: To zoom out right click or hold shift when using the zoom tool.

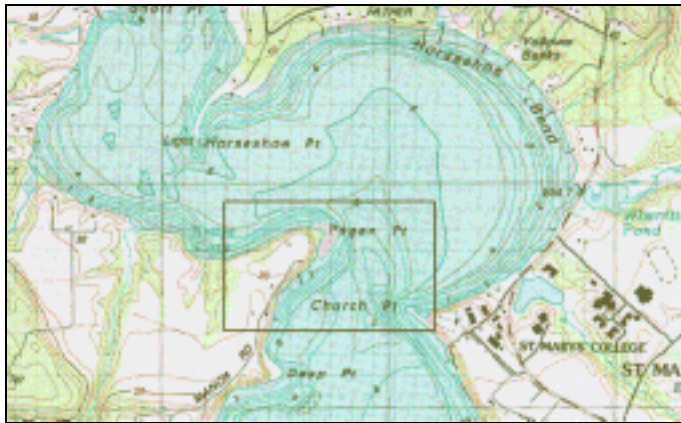



Figure 2-2 The main river portion of the image.

2.5 Using Feature Objects

A conceptual model is constructed over a background image using *feature objects* in the *Map*  module. Feature objects in *SMS* include points, nodes, arcs, and polygons, as shown in Figure 2-3. Feature objects are grouped into sets called *coverages*. Only one coverage is active at a time.

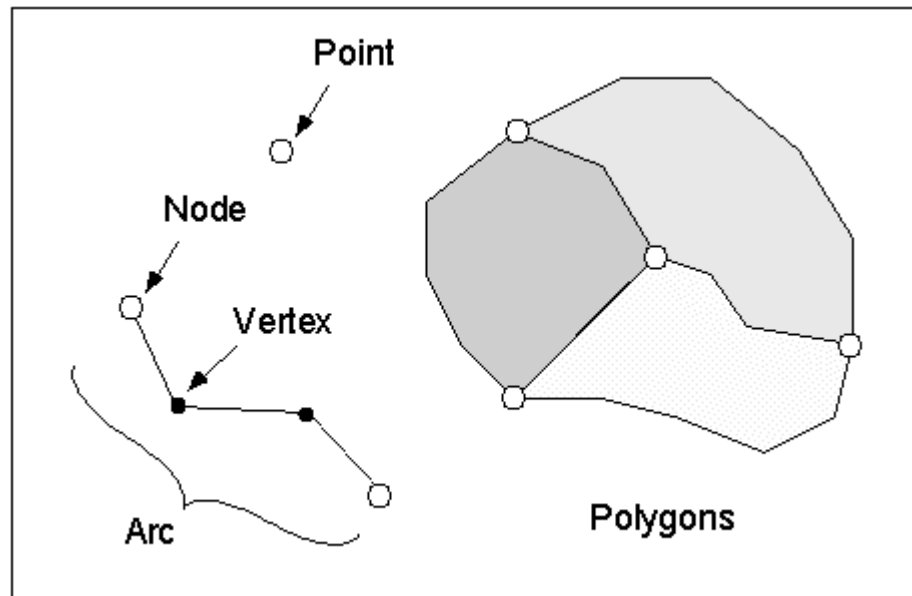


Figure 2-3 Feature Objects

A *feature point* defines an (x, y) location that is not attached to an arc. Points are used to force the creation of a mesh node at a specific location. A *feature node* is the same as a feature point, except that it is attached to at least one arc.

A *feature arc* is a sequence of line segments grouped together as a polyline entity. Arcs can form polygons or represent linear features such as channel edges. The two end points of an arc are called *feature nodes* and the intermediate points are called *feature vertices*.

A *feature polygon* is defined by a closed loop of feature arcs. A feature polygon can consist of a single feature arc or multiple feature arcs, as long as a closed loop is formed.

The conceptual model in this example will consist of a single coverage, in which the river regions and the flood bank will be defined. As you go along in this tutorial you will load new coverages over the existing coverage. The new coverage will become active and the old coverage will be inactive.

2.6 Creating Feature Arcs

A set of feature objects can be created to show topographically important features such as a river channel and material region boundaries. Feature objects can be digitized directly inside *SMS* or they can be converted from an existing *AutoCAD DXF* file. For this example, the feature objects will be digitized inside *SMS* using the registered TIFF image as a reference. To create the feature arcs:

1. Choose the *Create Feature Arc*  tool from the *Toolbox*.

2. Click out the left riverbank, as shown in Figure 2-4. As you create the arc, if you make a mistake and wish to back up, press the *BACKSPACE* key. If you wish to abort the arc and start over, press the *ESC* key. Double-click the last point to end the arc.



Figure 2-4 Creation of the first feature arc.

A feature arc has defined the general shape of the left riverbank. Three more arcs are required to define the right riverbank and the upstream and downstream river cross sections. Together, these arcs will be used to create a polygon that defines the study area. To create the remaining arcs:

- In the same manner just described, create the remaining three arcs, as shown in Figure 2-5. Remember to double-click the last point on each arc so that three separate arcs are created.



Figure 2-5 All feature arcs have been created.

You have now defined the main river channel. When creating your own models, you will proceed to create other arcs to separate material zones and define specific model features. To save time, the main features have been saved in a file. To open the file:

1. Select *File / Open*.
2. Open the file *stmary1.map* from the *tutorial\tut2* directory.

A new coverage is created from the data in the file, and the coverage you were using becomes inactive. The display should look like Figure 2-6.




Figure 2-6 The *stmary1.map* feature object data.

2.7 Manipulating Coverages

As stated at the beginning of this tutorial, feature objects are grouped into coverages. When a set of feature objects is opened from a file, a new coverage is created. The new coverage is made active, and the previous coverage becomes inactive. Inactive coverages are drawn in a mild green color by default.

When there are many coverages being drawn, the display can become cluttered. Individual coverages can be hidden to minimize this clutter. To hide all inactive coverages:


1. Select *Feature Objects / Coverages*. The active coverage is indicated with a check in the active column, while any visible coverages are indicated with a check in the visible column. The active coverage should be *stmary1.map*.
2. Click the *Hide All* button  to hide all the coverages.
3. Highlight the *stmary1.map* coverage and check the *Visible* option so that it is the only visible coverage.

4. Click the *OK* button to close the *Coverages* dialog.

As this tutorial progresses, you will be asked to open a few more map files. After doing so, you might want to hide or delete the old coverages so that the display does not become too cluttered.

2.8 Redistributing Vertices

To create the feature arcs, you simply clicked out a line of points on the image without paying attention to vertex distribution. The final element density in a mesh created from feature objects matches the density of vertices along the feature arcs, so it is desirable to have a more uniform node distribution. The vertices in a feature arc can be redistributed at a desired spacing. To redistribute vertices:

1. Choose the *Select Feature Arc*  tool from the *Toolbox*.
2. Click on the arc to the far right, labeled *Arc #1* in Figure 2-7.
3. Select *Feature Objects / Redistribute Vertices*. The *Redistribute Vertices* dialog shows information about the feature arc segments and vertex spacing.
4. Choose the *Specified spacing* option and enter a value of 470.
5. Click *OK* to redistribute the vertices along the arc.

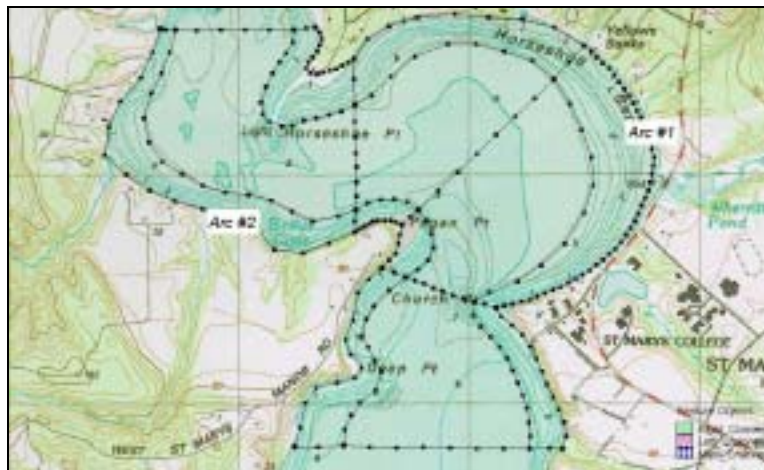


Figure 2-7 Redistribution of Vertices along arcs.

After clicking the *OK* button, the display will refresh, showing the specified vertex distribution. When you create your models, this would be done for each arc until you have the vertex spacing that you want. When you plan to use arcs in a patch, a better patch is created if opposite arcs have an equal number of vertices. In this case, you would want to use the *Number subdivisions* option rather than the *Specified spacing* option so that you can specify the exact number of vertices along each arc.

For this example, you should open another map file, which has the vertices redistributed on all the arcs. To open the map file:

- Open the file *stmary2.map*.

2.9 Defining Polygons

Polygons are created from a group of arcs that form a closed loop. Each polygon is used to define a specific material zone. Although polygons can be created one by one, it is faster to have *SMS* create them automatically. To have *SMS* build polygons out of the arcs:

1. Make sure no arcs are selected by clicking in the *Graphics Window* away from any arcs.
2. Select *Feature Objects / Clean* to be sure that there are no problems with the feature objects that were created. Click *OK* in the *Clean Options* dialog.
3. Select *Feature Objects / Build Polygons*.

Although nothing appears to have changed, polygons have been built from the arcs. The polygons in this example are for defining the material zones as well as to aid in creating a better quality mesh.

2.10 Assigning Meshing Parameters


With polygons, arcs and points created, *meshing parameters* can be assigned. These meshing parameters define which automatic mesh generation method will be used to create finite elements inside the polygon. For each method, a corner node will be created at each vertex on the feature arc. The difference comes in how internal nodes are created, and how those nodes are connected to form elements.

SMS has various mesh generation methods: patch, adaptive tessellation, paving, adaptive density and scalar paving density. These methods are described in the *SMS online Help*, so they will not be described in detail here. As an overview, adaptive tessellation is the default technique because it works for all polygon shapes, as does paving. Patches require either 3 or 4 polygonal sides. Density meshing options require scattered data sets to define the mesh density.

2.10.1 Creating a Refine Point For Adaptive Tessellation

When using the default Adaptive Tessellation method, some control can be maintained over how elements are created. A *refine point* is a feature point that is created inside the boundary of a polygon and assigned a size value. When the finite

element mesh is created, a corner node will be created at the location of the refine point and all element edges that touch the node will be the exact length specified by the refine point size value. To create a refine point:

1. Choose the *Select Feature Point*  tool from the *Toolbox*.
2. Double-click on the point inside the left polygon, labeled in Figure 2-8.
3. In the *Feature Point/Node Attributes* dialog, make sure the *Refine Point* option is on and enter a value of 100 (ft).
4. Click the *OK* button to accept the refine point.

When the finite element mesh is generated, a mesh corner node will be created at the refine point's location, and all attached element edges will be exactly 100 feet in length. A refine point is useful when a node needs to be placed at a specific feature, such as at a high or low elevation point.



Figure 2-8 The location of the refine point.

2.10.2 Defining a Coons Patch

As was previously stated, the *Coons Patch* mesh generation method requires exactly three or four sides to be created. However, very rarely do exactly three or four arcs make up a polygon. Figure 2-9 shows an example of a rectangular patch made up of four sides. Notes that *Side 1* and *Side 2* are both made from multiple feature arcs.

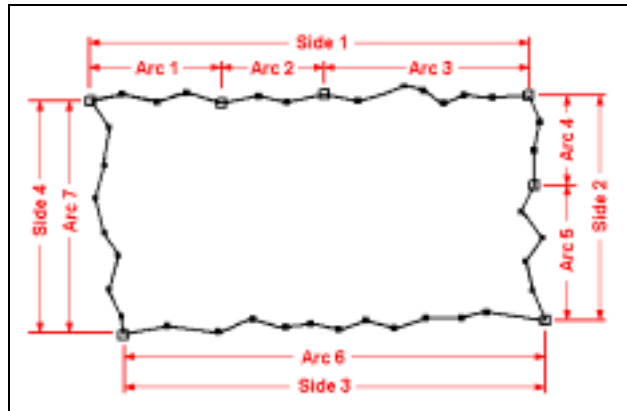



Figure 2-9 Four sides required for a rectangular patch.

SMS provides a way to define a patch from such a polygon by allowing multiple arcs to act as one. For example, the bottom middle polygon, labeled as *Patch Polygon*, contains five arcs, but it should be used to create a patch. To do this:

1. Choose the *Select Feature Polygon*  tool and double click on the bottom middle polygon.

In the middle of the *Feature Polygon Attributes* dialog, choose the *Select Feature Node*  tool.

Click on the node at the center of the left side, as seen in Figure 2-10.

Select the Merge option from the *Node Options* drop down list. This makes the two arcs on the left side be treated as a single arc. Notice that the *Patch* option may now be specified in the *Mesh Type* drop down list.

2. Select the *Patch* option from the *Mesh Type* drop down list. If you wish to preview the patch, click the *Preview* button.
3. Click the *OK* button to close the *Feature Polygon Attributes* dialog.

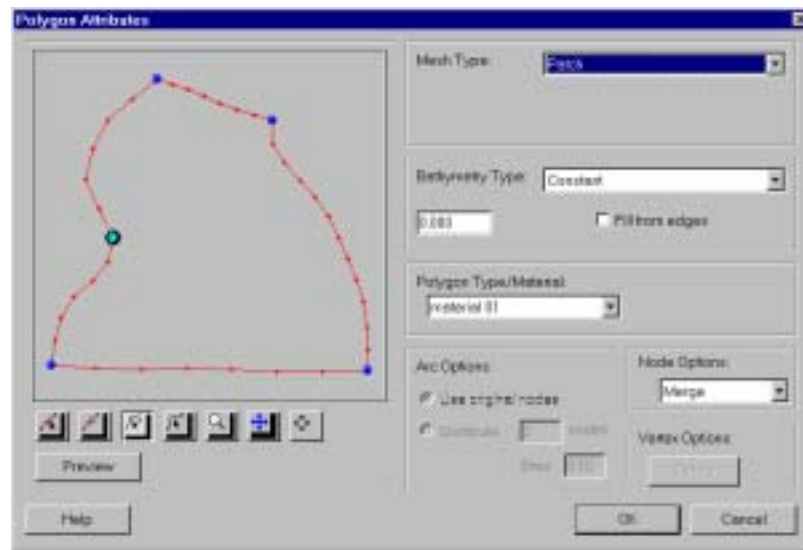


Figure 2-10 The Feature Polygon Attributes dialog.



When you are creating your models, you will need to set up the desired polygon attributes for each feature polygon in your model. Since this would take too much time for this tutorial, the rest of the polygons have been set up for you and saved to a map file. To import this data:

- Open the file *stmary3.map*.

In the coverage that opens, all polygon attributes have been assigned. The three main channel polygons are assigned as patches, while the other polygons are assigned as adaptive tessellation.

2.10.3 Removing Drawing Objects

Throughout this tutorial, *drawing objects*, such as labels and arrows, have been provided to give a description of certain feature objects. Drawing objects are not part of a coverage, so they do not become inactive. The drawing objects that have thus far been used will not be needed anymore. To delete the drawing objects:

1. Choose the *Select Drawing Objects*  tool from the *Toolbox*.
2. Choose *Edit / Select All* to select all drawing objects.
3. Press the *DELETE* key or click the *Delete*  macro from the *Toolbox*.
4. If *Edit / Confirm Deletions* is on click ok.

2.11 Applying Boundary Conditions

Before assigning boundary conditions, we must decide if we will use FESWMS or RMA2. To do this:

1. Select *Feature Objects / Coverages*.
2. Change the *Coverage type* to *TABS* (for RMA2) or *FESWMS*.
3. Push OK.

Boundary conditions can be assigned to arcs, points, and for *FESWMS*, polygons. Feature arcs may be assigned a flow, head, or flux status. Feature points may be assigned velocity or head values. Feature polygons may be assigned ceiling elevation functions, but only in a *FESWMS* coverage.

The inflow for this example is across the top of the model and the outflow is across the bottom. Notice that there are three feature arcs across each of these sections. A flow rate value could be assigned to each of the arcs at the inflow. However, this would create three separate inflow nodestrings, connected end-to-end. The same situation exists at the outflow cross section.

To avoid creating three separate boundary conditions at a single cross section, an *arc group* can be defined. An arc group consists of multiple arcs that are linked together. The arc group can be assigned the boundary condition instead of assigning it at the individual arcs so that when the model is generated, only a single nodestring is created, which spans the entire cross section.

2.11.1 Defining Arc Groups

For this example, two arc groups will be defined, one at the inflow boundary and one at the outflow boundary. To create the arc groups:



1. Choose the *Select Feature Arc*  tool from the *Toolbox*.
2. Holding the *SHIFT* key, select the three arcs that make up the flow cross-section, labeled as *Flow Arcs* in Figure 2-11.
3. Select *Feature Objects / Create Arc Group* to create an arc group from the three selected arcs.
4. Now, select the three arcs that make up the head cross-section, labeled as *Head Arcs* in Figure 2-11.
5. Select *Feature Objects / Create Arc Group*.



Figure 2-11 The arc groups to create.

2.11.2 Assigning The Boundary Conditions

With the arc groups created, boundary conditions can now be assigned. To assign the inflow boundary condition:

1. Choose the *Select Arc Group*  tool from the *Toolbox*.
2. Double-click the arc group at the inflow (top) cross section.
3. In the *Arc Group Attributes* dialog, select the *Boundary Conditions* option, and click the *Options* button.
4. Change to *Flow BC* and assign a *Flowrate* of 40,000 cfs.
5. Click the *Perpendicular to boundary* button to force the flow to enter the mesh perpendicular to the inflow boundary.
6. Click the OK button in both dialogs.


To assign the water surface boundary condition:

1. Double-click the arc group at the outflow cross section.
2. In the *Arc Group Attributes* dialog, select the *Boundary Conditions* option, and click the *Options* button.
3. Click the *Head BC* button and assign an *Elevation* (water surface) of 20 ft.
4. Click the OK button in both dialogs.

The inflow and outflow boundary conditions are now defined in the conceptual model. When the conceptual model is converted to a finite element mesh, *SMS* will create the nodestrings and assign the proper boundary conditions.

2.12 Assigning Materials To Polygons

Each polygon is assigned a material type. All elements generated inside the polygon are assigned the material type defined in the polygon. To assign the materials:

1. Choose the *Select Feature Polygon*  tool from the *Toolbox*.
2. Double-click on any of the polygons.
3. In the *Feature Polygon Attributes* dialog, make sure the *Polygon Type/Material* shows the correct material for the polygon, as shown in Figure 2-12.
4. Click the *OK* button to close the *Feature Polygon Attributes* dialog.

Repeat these steps to make sure the correct material type is assigned to each of the feature polygons.

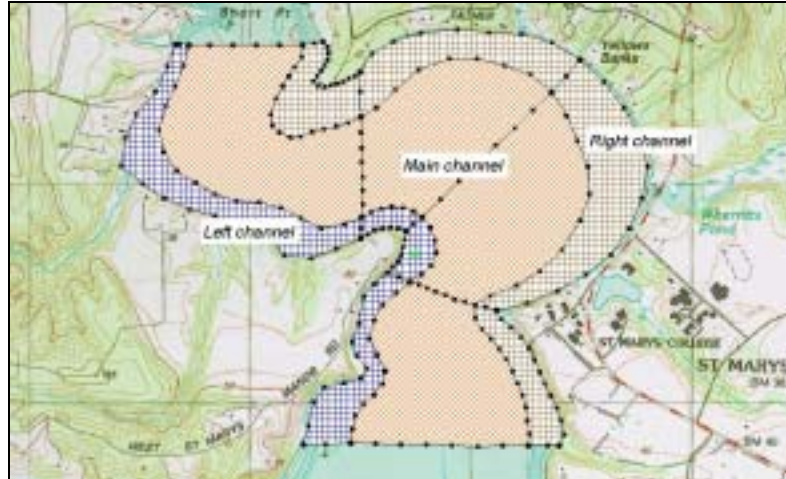


Figure 2-12 Polygons with defined material types.

2.12.1 Displaying Material Types

With the materials assigned to the polygons, you can fill the polygons with the material colors and patterns. To do this:

1. Click the *Display Options*  macro from the *Toolbox*.

2. If not active, select the *Map* tab in the *Display Options* dialog.
3. Turn on the *Polygon Fill* option and select the *Fill with materials* option.
4. Click the OK button to close the *Display Options* dialog.


The display will refresh, filling each polygon with the material color and pattern.

2.13 Converting Feature Objects To a Mesh

With the meshing techniques chosen, boundary conditions assigned, and materials assigned, we are ready to generate the finite element mesh. To do this:

1. Select *Feature Objects / Map -> 2D Mesh*.
2. Click the *OK* button to start the meshing process.

After a few moments, the display will refresh to show the finite element mesh that was generated according to the preset conditions. With the mesh created it is often desirable to delete or hide the feature arcs and the image. To do this:

1. Click the *Display Options*  macro from the *Toolbox*.
2. If it is not active, select the *Map* tab.
3. Turn off the display of *Arcs*, *Nodes*, and *Polygons*.
4. Turn off the *Display image* option.
5. Click the *OK* button to close the *Display Options* dialog.

The display will refresh to show the finite element mesh, as shown in Figure 2-13. With the feature objects and image hidden, the mesh can be manipulated without interference, but they are still available if mesh reconstruction is desired.

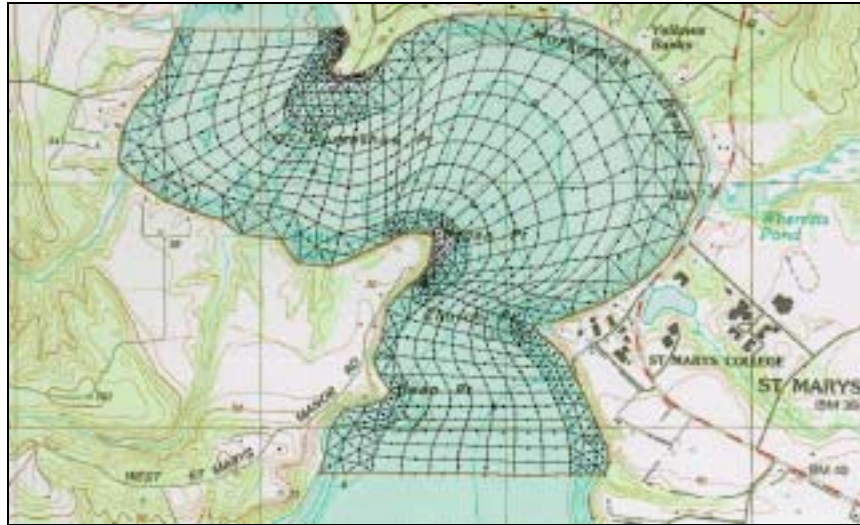



Figure 2-13 The finite element mesh that was created.

2.14 Editing the Feature Object Mesh

When a finite element mesh is generated from feature objects, it is not always the way you want it. An easy way to edit the mesh is to change the meshing parameters in the conceptual model, such as the distribution of vertices on feature arcs or the mesh generation parameters. Then, the mesh can be regenerated according to the new parameters. If there are only a few changes desired, they can be edited manually using tools in the mesh module. These tools are described in the *SMS Help* in the section on the *Mesh Module*.

2.15 Interpolating To The Mesh

The finite element mesh generated from the feature objects defines only the X- and Y- coordinates for the nodes. To get the bathymetric information, survey data saved as scatter points can be interpolated onto the finite element mesh. To open the scattered data:


1. Switch to the *Scatter Point*  module.
2. Select *File / Open* and open the file *stmaryscat.sup*.

The screen will refresh, showing a set of scattered data points. Each point represents a survey measurement. Scatter points are used to interpolate bathymetric (or other) data onto a finite element mesh. Although this next step requires you to manually interpolate the scattered data, this interpolation can be set up to automatically take place during the meshing process. To interpolate the scattered data onto the mesh:

1. Select *Scatter / Interpolate to Mesh*.
2. In the *Interpolate To Mesh* dialog, select *Linear* from the *Interpolation* drop down list. (For more information on *SMS* interpolation options, see the *SMS Online Help*.)
3. Turn on the *Map Elevations* option at the lower left area of the dialog.
4. Click the *OK* button to perform the interpolation.



The scattered data is triangulated when it is read into *SMS* and an interpolated value is assigned to each node in the mesh. The *Map Elevations* option causes the newly interpolated value to be used as the nodal *Z*- coordinate.

As with the feature objects, the scattered data will no longer be needed and may be hidden or deleted. To hide the scatter point data:

1. Click the *Display Options*  macro from the *Toolbox*.
2. Select the *Scatter* tab if it is not already active.
3. Turn off the *Visible* option for the scatter set and push *OK*.

2.16 Renumbering the Mesh

The process of creating and editing a finite element mesh can cause the node and element ordering to become disorganized. Renumbering the mesh can restore a good mesh ordering. (The mesh is renumbered after the mesh generation, but the mesh is renumbered from an arbitrary nodestring, which does not always give the best renumbering). To renumber:

1. Switch to the *Mesh*  module.
2. Choose the *Select Nodestring* tool  from the *Toolbox*.
3. Select the flow nodestring at the top of the mesh by clicking inside the icon that is at the middle of the nodestring.
4. Select *Nodestrings / Renumber*. Leave the renumber method at *Band Width* and push *OK*.

2.17 Saving a Project File

Much data has been opened and changed, but nothing has been saved yet. The data can all be saved in a project file. When a project file is saved, several files are saved. Separate files are created for the map, scatter, and mesh data. The project file is a text file that references the individual data files. To save all this data for use in a later session:

1. Select *File / Save Project*.
2. Save the file as *stmaryout.spr*.
3. Click the *Save* button to save the files.

2.18 Conclusion

This concludes the Overview tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File / Exit*. If prompted to confirm, click the *Yes* button.

Mesh Editing

This tutorial lesson teaches manual finite element mesh generation techniques that can be performed using *SMS*. It gives a brief introduction to tools in *SMS* that are useful for editing a finite element mesh. The mesh in this tutorial will be created by hand from survey points. These mesh editing methods should be used in conjunction with map module meshing to generate a good finite element mesh. This tutorial exists to show useful tools for editing small portions of a mesh after the mesh generation. All files for this tutorial are in the *tutorial\tut3* directory.

3.1 Importing Topographic Data

Data points for a finite element mesh can be generated directly from topographic data, such as a list of survey points. An *XYZ* file contains the header *XYZ* on the first line of the file and then the *X*, *Y*, and *Z* coordinates of each point on a single line in the file. This type of file can be opened by *SMS*. To open the *poway1.xyz* file:

1. Select *File | Open*.
2. Change to the *tutorial\tut3* directory and open the file *poway1.xyz*.
3. Change the *Open file as* to *Mesh* click *Next >*, uncheck the triangulate data option, and then click *Finish*.

The data points from the file are converted to *mesh nodes*. From the *File Import Wizard*, a user can open any columnar data into *SMS*. By default, the columns are set up as *x*, *y* and *z*. See the *SMS online Help* for more information on the import wizard. The data points created from *poway1.xyz* are shown in Figure 3-1.

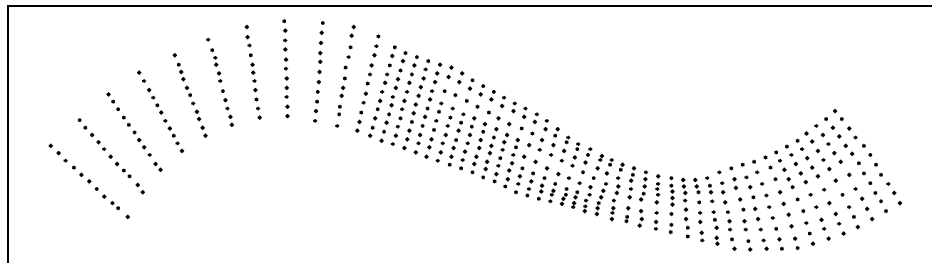


Figure 3-1 The poway1.xyz data points.

3.2 Triangulating the Nodes

After nodes have been created, elements are required to build a finite element mesh. Elements connect the nodes to define the extents of the flow area. SMS provides numerous automatic mesh generation techniques. This section will review a very simple technique, *triangulation*. If the *triangulate data* option had not been unchecked above, this step would have been done automatically when the file was imported. The file would then have looked like Figure 3-2 when it was opened. To create a triangulated mesh from the data points:

1. Select *Elements / Triangulate*.
2. Since no nodes are selected, you will be prompted to triangulate all of them. Click the *Yes* button at this prompt.

When SMS triangulates data points, it creates either quadratic triangles or linear triangles from the mesh nodes. Different numerical models support different types of elements. *RMA2*, *FESWMS*, and *RMA10* support quadratic elements, while *HIVEL*, *ADCIRC*, and *CGWAVE* support only linear elements. After the nodes are triangulated, the mesh will look like that in Figure 3-2. It may or may not have midside nodes, depending on whether the elements are linear or quadratic.

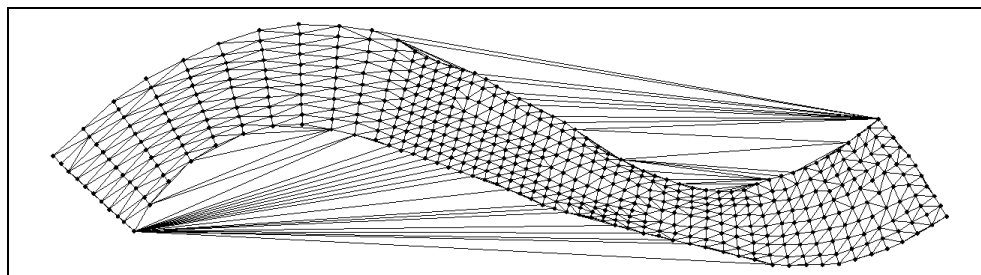




Figure 3-2. The results of triangulating the poway1.xyz data.

3.3 Deleting Outer Elements

The triangulation process always creates elements outside the real mesh boundaries. For this tutorial, the mesh should be in the shape of a rotated *S*, so any elements outside of this boundary must be deleted. To remove these elements:

1. Choose the *Select Elements* tool  from the *Toolbox*.
2. Click on an element to select it.
3. Select another element by holding the *SHIFT* key and clicking on it.
4. Select *Edit | Delete* or press the *DELETE* key to remove the selected elements.

It is tedious to individually select every element that needs to be deleted. *SMS* provides a hot key to help selecting groups of adjacent elements. To select a group of adjacent elements:

1. Choose the *Select Elements* tool  from the *Toolbox*.
2. Hold the *CTRL* key.
3. Click and drag a line through some elements to select them. Be careful to only select elements outside the *S* shape.
4. Select *Edit / Delete* or press the *DELETE* key to remove the selected elements.

Continue deleting elements that are outside the boundaries of the *S* shape.

3.4 Deleting Thin Triangles

Many times, triangulation creates very thin triangular elements outside the desired mesh boundary. The three corner nodes of thin triangles are almost collinear and the elements may be too thin to see or select. If these are not deleted, large errors in the model solution can result.

SMS provides a way to define what is meant by a thin triangle using the element *aspect ratio*. The element aspect ratio is the ratio of the element width to its height. Perfect equilateral triangles have an aspect ratio of 1.0 while that of thin triangles is much less. To define the element aspect ratio:

1. Select *Elements | Options*.

2. Set the aspect ratio in the *Select thin triangle aspect ratio* box to 0.1. (The default value is 0.04). Triangular elements with an aspect ratio less than this are considered to be thin triangles.
3. Click the *OK* button.

The best aspect ratio to use for selecting thin triangles depends on the finite element mesh. For this mesh, the distribution of nodes is rather uniform, so a large aspect ratio will suffice. After this value is set, *SMS* can check for and select thin triangles. To delete any remaining thin triangles:

1. Select *Elements / Select Thin Triangles*. The lower right portion of the *Status Bar* in the *Graphics Window* shows how many elements became selected due to this operation, along with the total area of the selected elements. There may be quite a few elements selected.
2. Select *Edit / Delete* or press the *DELETE* key. If prompted to confirm, click the *Yes* button.

The mesh should now look like Figure 3-3.

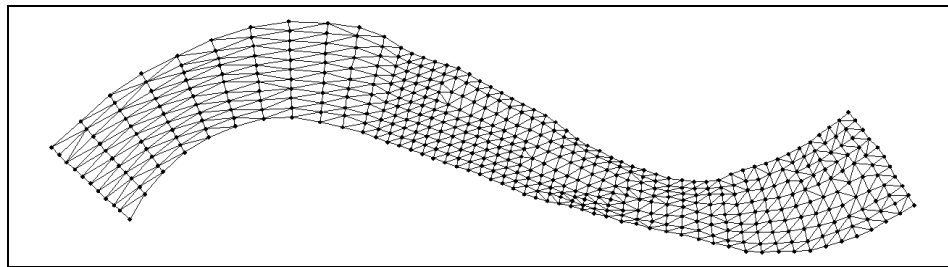


Figure 3-3. The *poway1* mesh after deleting excess triangles.

3.5 Merging Triangles

The mesh is composed entirely of triangles. Both *ADCIRC* and *CGWAVE* support only triangles. If you are using one of these models, you may skip this section of this tutorial.

Quadrilateral elements are generally preferred when using *RMA2*, *RMA10*, *FESWMS*, or *HIVEL* because:

- They make a more concise mesh for faster solutions.
- Quadrilateral elements are numerically more stable.

SMS can automatically merge a pair of triangles into a quadrilateral. Before merging triangles, the *Merge triangles feature angle* should be set. To do this:

1. Select *Elements / Options*.
2. Enter a value of 55.0 in the *Merge triangles feature angle* box (the default value is 65.0). Two triangles may be merged if all angles of the resulting quadrilateral are greater than the value specified.
3. Click the *OK* button.

The finite element method is more stable and accurate when quadrilateral elements are rectangular and triangular elements are equilateral. Although it is not practical for a mesh to exist entirely of these perfect shapes, the elements should approach these shapes as close as possible. For this reason, *SMS* merges triangles in an iterative manner. First, it merges elements using the angle criterion of 90°. Then, the angle criterion is decreased by a number of steps to the feature angle specified. Slowly decreasing the feature angle and testing all triangles against this will form the best-shaped elements.

SMS can merge the triangles in either a selected portion of elements or all elements. In order to merge triangles in the entire mesh, no elements should be selected. To merge triangular elements into quadrilateral elements:

1. Select *Elements / Merge Triangles*.
2. Since no elements are selected, you will be prompted to merge all triangles. Click the *Yes* button at this prompt.

With most meshes, as is the case for this example, not all triangles will be merged. The mesh will appear as in Figure 3-4 after *SMS* merges the triangles.

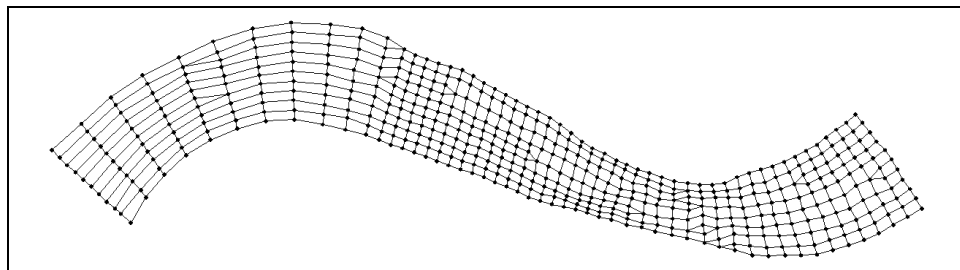




Figure 3-4. The poway1 mesh after merging triangles.


3.6 Editing Individual Elements

After triangulating the nodes, deleting elements outside the boundaries, and merging triangles, the mesh often needs further manipulation to add model stability. For a main river channel such as this model, lines of elements should run parallel to the mesh boundary. This is especially important in cases where a portion of the mesh may become dry so that the mesh will dry parallel to the boundary. Two of the tools in *SMS* used for manipulating individual elements are the *Split / Merge*  tool and

Swap Edge  tool. With the *Split / Merge* tool, two adjacent triangular elements can be merged into a quadrilateral element or a single quadrilateral element can be split into two triangular elements. With the *Swap Edge* tool, the common edge of two adjacent triangular elements can be swapped. See the *SMS Help* for a better description of these tools.

3.6.1 Using the Split / Merge Tool

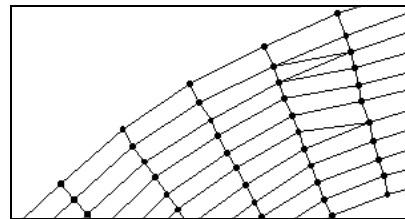
Most triangular elements in this mesh were merged into quadrilateral elements when the *Merge Triangles* command was performed in section 3.5. Some of the elements that were not automatically merged can be merged manually. To do this:

1. Zoom into the portion of the mesh shown in Figure 3-5a. Notice the two triangular elements separated by a number of quadrilateral elements.
2. Select the *Split / Merge* tool  from the *Toolbox*.
3. Split the quadrilateral, highlighted in Figure 3-5a, by clicking inside it. There should now be three triangles, as shown in Figure 3-5b.
4. Merge the top two triangles, highlighted in Figure 3-5c, by clicking on the edge between them. (The *Split / Merge* tool should still be selected.)

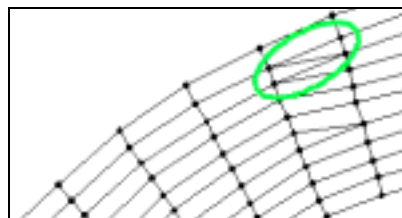
The result of this split/merge operation is shown in Figure 3-5d. There is now one fewer quadrilateral between the two triangles.



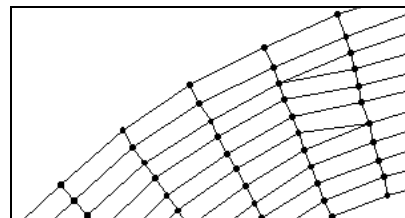
(a). Initial elements.



(b). After splitting quadrilateral.



(c). After swapping edge.



(d). Final elements.

Figure 3-5. Example of manual split / merge procedure.

To finish editing this section:

- Repeat the above split/merge process until there are no more triangles across the section. This part of the mesh should look like Figure 3-6.

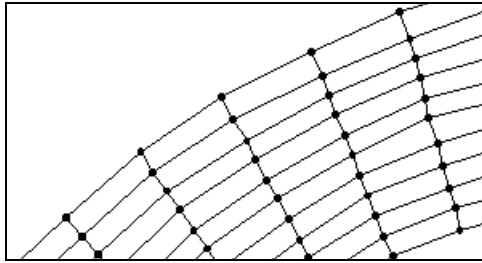


Figure 3-6. The mesh section after merging triangles.

3.6.2 Using the Swap Edge tool

The common edge between two triangles can be swapped. The best way to understand this is to think of the two triangles as a quadrilateral, and the common edge between them is a diagonal of the quadrilateral. By swapping this common edge, it changes to be along the opposite diagonal of the quadrilateral. If this edge is clicked again, it returns back to its original state. This can be seen in Figure 3-7.

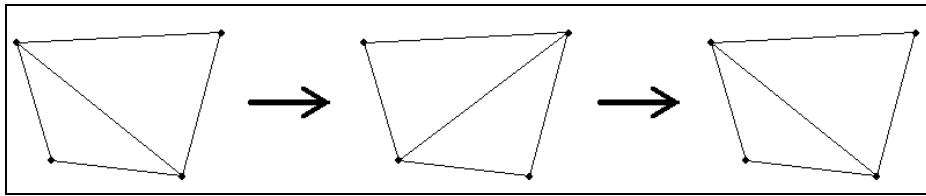




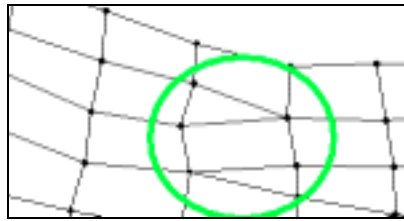
Figure 3-7 The Swap Edge technique.

One place in this mesh requires the use of the *Swap Edge* tool together with the *Split / Merge* tool to be able to merge the triangles. This is located toward the middle of the finite element mesh, at the constriction. The easiest way to find this location is to set the window boundaries to the correct location. To do this:

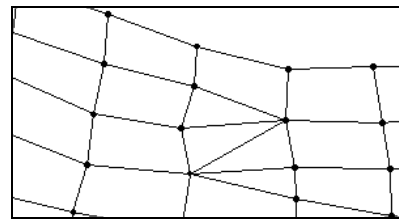
1. Select *Display / View | Window Bounds*.
2. In the *Set Window Boundaries* dialog, select to use the *X range to be specified* option (the *Y at top* field will become disabled).
3. Enter these values: *X at left* = 25,200; *X at right* = 25,500; *Y at bottom* = 9300.
4. Press the *OK* button.

You should now be able to see the portion of the finite element mesh shown in Figure 3-8. In this part of the mesh, there are two triangles that need to be merged together, separated by a single quadrilateral. To do this:

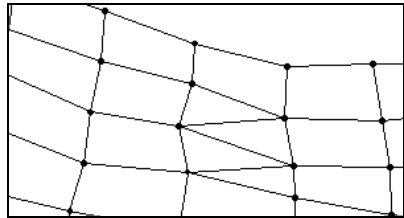
1. Choose the *Split / Merge*  tool from the *Toolbox*.
2. Click inside the quadrilateral, highlighted in Figure 3-8a, that separates the two triangles. The quadrilateral gets split as shown in Figure 3-8b. The new edge was not created in the direction necessary to merge the outer triangles.
3. Choose the *Swap Edge*  tool from the *Toolbox*.
4. Click only once, directly on the edge that was just created inside the quadrilateral. The edge will swap to the other diagonal of the quadrilateral. This result is shown in Figure 3-8c.
5. Once again choose the *Split / Merge* tool from the *Toolbox*.
6. Merge the top two triangles to form one quadrilateral, and then merge the bottom two triangles to form another quadrilateral. The result is shown in Figure 3-8d.



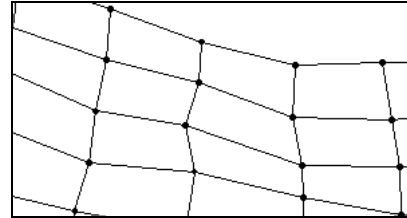
(a). The original elements.



(b). Elements after splitting quad.



(c). Elements after swapping edge.



(d). Final quadrilateral elements.

Figure 3-8 Example of manual swapping procedure.

Although this operation appears simple, it is one that takes some time to get used to performing. Most people do not get through this without making a mistake. However, after you understand this operation, it is easier to use. The *Split / Merge* and *Swap Edge* tools are very useful for manually adjusting small areas of the finite element mesh.

Continue to merge triangles in the areas that you are able to do so. Not all of the triangles can be merged. When you are done, there should be only six triangles left in the finite element mesh, and it should look like that shown in Figure 3-9.

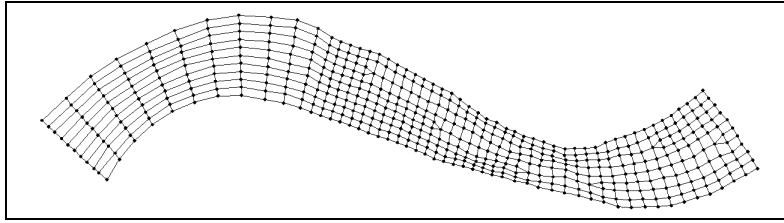



Figure 3-9 The finite element mesh after merging triangles.

3.7 Smoothing the Boundary

When dealing with quadratic finite element meshes, mass loss can occur through a jagged boundary. It is good to smooth the boundary of a quadratic mesh to prevent these losses. Smoothing can only be performed with quadratic models, because the midside nodes get moved while corner nodes do not. *SMS* currently supports three quadratic finite element models, *RMA2*, *RMA10*, and *FLO2DH*. If you are not using one of these quadratic models, you can skip this section. The quadratic models still support the creation of linear elements. To make sure you have quadratic elements:

1. Select *File / Get Info*.
2. In the top right corner of the *Mesh Information* dialog, look at the *Element type* defined. This will be either *quadratic* or *linear*.
3. Click the *Close* button in the *Mesh Information* dialog.
4. If the element type was *linear*, select *Elements / Linear <-> Quadratic* to switch the element type. If it was *quadratic*, you are already set.

The easiest way to smooth the entire mesh boundary is by creating a nodestring around the entire mesh boundary. To do this:

1. Choose the *Create Nodestring*  tool from the *Toolbox*.
2. Click the node labeled *Node 1* in Figure 3-10.
3. Hold the *CTRL* key and double-click the node labeled *Node 2* in Figure 3-10. When holding the *CTRL* key, *SMS* creates a nodestring counter-clockwise around the mesh boundary from the first node to the second node.

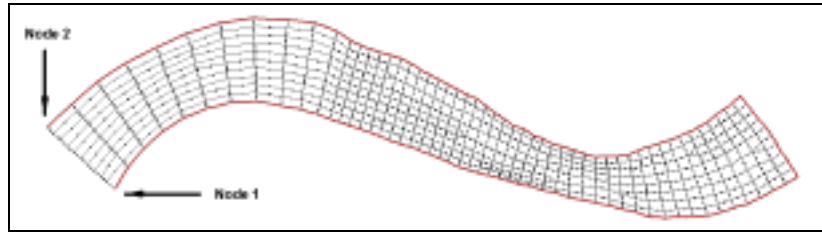



Figure 3-10 The nodestring to create for smoothing.

This nodestring starts from *Node 1*, and runs counter clockwise around the entire boundary to *Node 2*. Notice that this nodestring goes around two sharp corners on the right side of the mesh. To assure that these corners remain sharp:

1. Select *Elements / Options*.
2. In the *Element Options* dialog, change the *Smooth nodestring feature angle* to be 45.0. A midside node will not move if it is at a corner that is sharper than this angle.
3. Click the *OK* button.

Now that the nodestring is created and the feature angle is set, the boundary is ready to be smoothed. To do this:

1. Choose the *Select Nodestrings*  tool from the *Toolbox*. A small icon will appear at the center of the nodestring.
2. Click on the icon to select the nodestring. The icon will be filled and the nodestring will be highlighted in red.
3. Select *Nodestrings / Smooth*. The mesh boundary will be smoothed as shown in Figure 3-11.

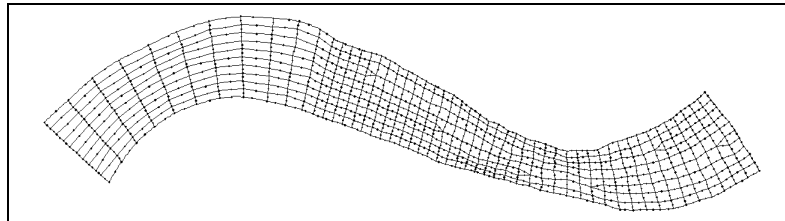




Figure 3-11. Example nodestrings for smoothing the mesh boundary.

In general, it is sufficient to smooth the finite element mesh boundary. However, it may be desirable to further smooth interior elements at sharp bends or where dry elements may change the boundary. Any nodestring can be used for smoothing. See the *SMS Help* for more information on creating interior nodestrings and the smoothing operation.

3.8 Renumbering the Mesh

The process of creating and editing a finite element mesh, as performed in the previous few sections, causes the node and element ordering to become disorganized. This random mesh ordering increases the size of the matrices required by the finite element analysis codes. Renumbering the mesh restores a good mesh ordering, making it faster to run the analysis. Renumbering starts from a nodestring. To renumber this mesh:

1. Choose the *Create Nodestring* tool  from the *Toolbox*.
2. Create a nodestring across the left section, as shown in Figure 3-12.
3. Choose the *Select Nodestring* tool  from the *Toolbox* and select the nodestring that was just created.
4. Choose *Nodestrings / Renumber*. You can choose either the *Band Width* or *Front Width* option for this mesh. For more information on these two options, see the *SMS Help*.
5. Click the *OK* button to start the renumbering process.

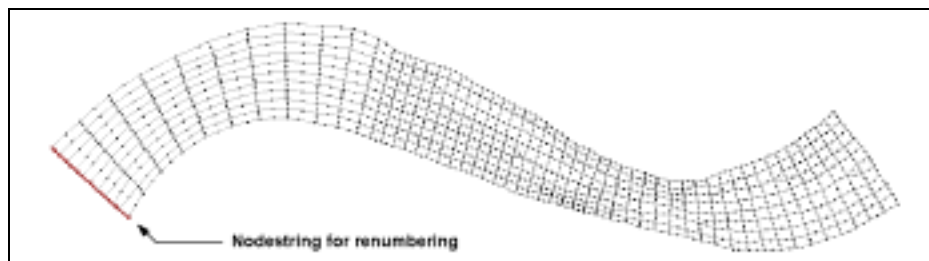



Figure 3-12. The position of the nodestring for renumbering.


When *SMS* is finished renumbering the mesh, the display will refresh. Remember that adding and deleting nodes or elements changes the mesh order. It is important that renumbering be the last step of the mesh creation process. Editing a mesh invalidates any boundary condition and/or solution files that have previously been saved. (Boundary condition and solution files are discussed in later tutorials).

3.9 Changing the Contour Options

When the finite element mesh is created, contours lines are drawn to connect points of equal elevation. By default, these contours are displayed as constant green lines. The contour display can be changed using the *Contour Options* dialog. It is always a good idea to look at a color contour map after a new finite element mesh has been created. This helps you better visualize the bathymetry of the model. To set the color fill contours:

1. Choose *Data / Contour Options* or click the *Contour Options*  macro.
2. In the *Contour Method* section of the *Contour Options* dialog, select the *Color fill* option from the drop down list.
3. Click the *Options* button for the contour color, set the Palette method to *Hue Ramp*, and click OK.
4. Click the *OK* button.

The display will refresh with color filled contours such as those shown in Figure 3-13. If you do not see color filled contours, then the display of contours has been turned off. To turn the contour display back on:

1. Choose *Display / Display Options* or click the *Display Options*  macro.
2. Select the *2D-Mesh* tab if it is not already selected.
3. Make sure the *Contours* option is checked.
4. Select the *Contours* tab of the *Display Options*.
5. Change the *Contour Method* drop down box to *Color Fill*.
6. Click the *OK* button.

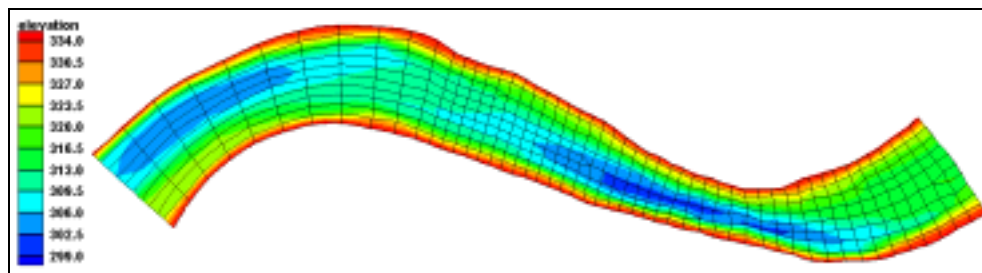



Figure 3-13 Elevation contours of the poway1 mesh.

In this plot, you can see that there are two pits in the river, while both banks are the highest part. For more examples of how to work with display and contour options in SMS, see the *SMS Help*.

3.10 Checking the Mesh Quality

Another important thing to check with a newly created finite element mesh is the mesh quality. There are various things that SMS looks at when checking this. To turn on the mesh quality:

1. Select *Display / Display Options* or click the *Display Options*  macro.
2. Select the *2D-Mesh* tab if it is not already selected.
3. Uncheck the *Contours* box.
4. Check the *Mesh quality* box.
5. Click the *OK* button.

The display will refresh without contours and with the mesh quality, as shown in Figure 3-14. The mesh quality shows where problem areas may occur. A legend shows the color corresponding with each quality item. See the *SMS Help* for more information on these mesh quality options.

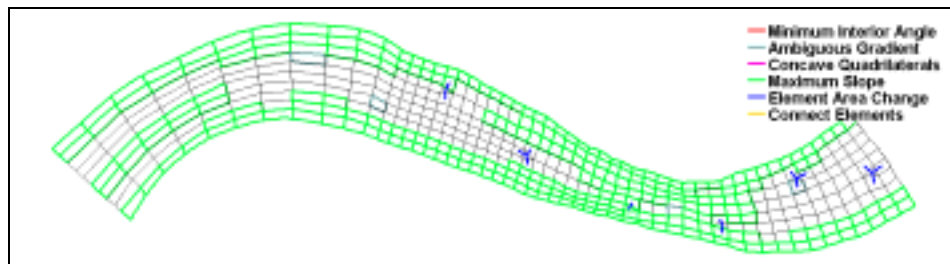



Figure 3-14 Mesh quality for the Poway1 finite element mesh.

Many elements are highlighted because of the maximum slope warning. Elements that are steep in the flow direction may cause supercritical flow to occur. In this mesh, however, the elements are steep in the direction perpendicular to the flow, so this warning can be ignored. To turn off this mesh quality check:

1. Select *Display / Display Options* or click the *Display Options*  macro from the *Toolbox*.
2. In the *Mesh Display Options* dialog, click the *Options* button next to the *Mesh quality* item.
3. In the *Element Quality Checks* dialog, turn off the *Maximum slope* option.
4. Click the *OK* button in both dialogs.

Once again, the display will refresh (see Figure 3-15), but this time, no slope warnings will be shown. There are only two warning types that remain. The *Element Area Change* warning is not crucial, especially when there are not many of these warnings in the mesh. For this mesh, this element quality warning will be ignored. The *Ambiguous Gradient* warning is shown for four elements, which are numbered in the figure. If the flow through these elements is deep, then the ambiguous gradient will not really affect the flow pattern. However, if the flow through these elements is

shallow, the element can become an artificial dam. Since the flow depths are not yet known for this finite element mesh, these quality problems will be fixed.

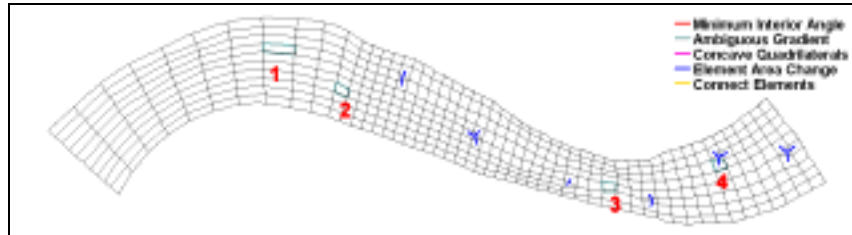

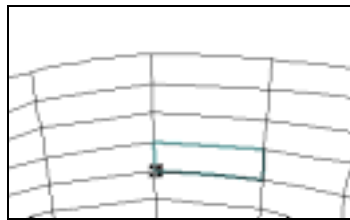


Figure 3-15 Mesh Quality without the Maximum Slope quality check.

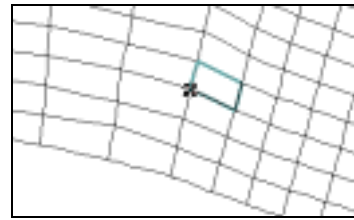
If the ambiguous gradient is not very large, it can be fixed by slightly modifying nodal elevations. This is the case with elements 1, 2, and 4 in Figure 3-16. A large ambiguous gradient, which would require editing nodal elevations by more than a foot or two, should just be split into two triangular elements. This is the case with element 3 in Figure 3-16.

The ambiguous gradients can be removed. To do this:

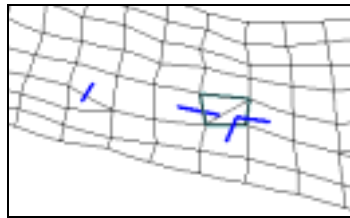
1. Choose the *Select Nodes*  tool from the *Toolbox*.
2. For element 1, select the bottom left corner node, as shown in Figure 3-16a. Increase its elevation by 0.5 feet by typing the value in the *Edit Window*. Press *ENTER* to accept the change.
3. For element 2, select the bottom left corner node, as shown in Figure 3-16b. Increase its elevation by 0.2 feet.
4. For element 3, split and swap the quadrilateral as necessary so that it looks like Figure 3-16c.
5. For element 4, select the upper right corner node, as shown in Figure 3-16d. Increase its elevation by 0.2 feet.



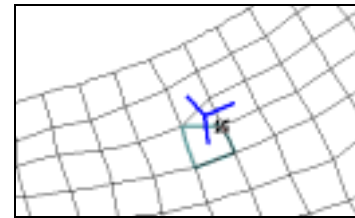
(a). Select node on element 1.



(b). Select node on element 2.



(c). Split element 3.



(d). Select node on element 4.

Figure 3-16 The four quadrilateral elements with an ambiguous gradient.

After making these modifications, you will not have any more ambiguous gradient warnings in the finite element mesh. The following three things should now be done (in no particular order):


- *Turn off the display of element quality checks.* You are done looking at the mesh quality, so this should be turned off to make the screen less busy.
- *Turn on the display of color filled contours.* Check that the adjustments you made did not make funny looking contours in the mesh. When editing nodal elevation values, it is always important to check the contours. If funny looking contours result, you may want to put things back the way they were and make some different changes.
- *Renumber the mesh.* Remember, whenever you adjust the finite element mesh, it should be renumbered. If you had only modified elevation values, then the mesh would not require renumbering. However, you must renumber the mesh after splitting a quadrilateral into two triangles.

3.11 Refining Elements


At times, it is desirable to refine part of a mesh so that there is more definition in that area. More definition helps to increase accuracy and decrease divergence problems. It is important to not refine too much of a mesh, however, because more nodes and elements increase the time required for finite element computations. In this section, you will refine the section of elements on the left edge of the mesh.

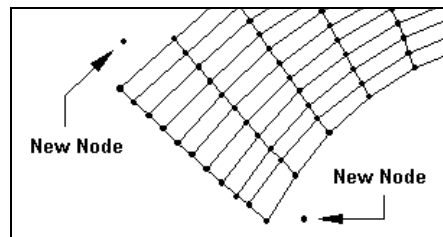
3.11.1 Inserting Breaklines

The elements at the left of the mesh are already rather skinny. The first refinement will be to cut them across their width. This can be done using a nodestring as a breakline. As you have seen previously, a nodestring must be created using existing nodes. Therefore, before being able to create a nodestring to cut the elements, you must create two nodes, one on either side of the channel. To do this:

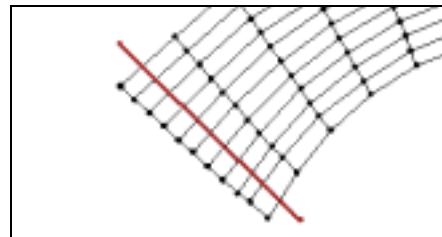
1. Choose the *Create Nodes*  tool from the *Toolbox*.
2. Click once on each side of the channel, near the middle of the left-most column of elements, as shown in Figure 3-17a.

A nodestring can now be created from one of these new nodes to the other. This nodestring will be used as a breakline. To create the nodestring:

1. Choose the *Create Nodestring*  tool from the *Toolbox*.
2. Click on one of the new nodes. Double-click on the other. The nodestring will appear, as shown in Figure 3-17b.




(a). Two nodes to create.



(b). The nodestring to create.

Figure 3-17 Adding the nodestring to use as a breakline.

With the nodestring created, it can be used as a breakline. A breakline splits all the elements that it crosses, forcing element edges to appear along the line. To make a breakline from the nodestring:

1. Choose the *Select Nodestrings*  tool from the *Toolbox*.
2. Select the icon that appears in the center of the nodestring.
3. Select *Nodestrings / Force Breaklines*. The elements will be split along the nodestring, as shown in Figure 3-18.

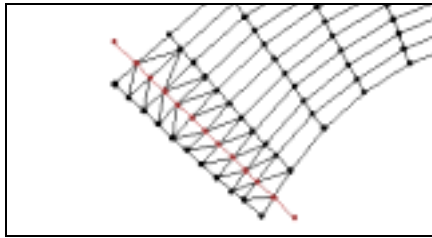




Figure 3-18 The breakline has been inserted.


Now that the nodestring has been used as a breakline, it is no longer needed. It should still be selected. To remove the nodestring:

- Select *Edit / Delete* or click the *Delete*  macro.

When the elements get broken along the breakline, triangular elements are created. These should be merged into quadrilateral elements. To do this:

1. Choose the *Select Elements*  tool from the *Toolbox*.
2. Select *Edit / Select With Poly*. This allows you to select a specific set of elements by drawing a polygon around them.
3. Click out a polygon that surrounds all the triangular elements that were created by the breakline. Double-click to end the polygon.
4. With the triangular elements highlighted, select *Elements / Merge Triangles*.

All of the triangular elements that were created by the breakline will be merged into quadrilateral elements. With these elements created, you just need to get rid of the two nodes that were created to define the breakline. These nodes are not connected to any elements, and are thus called *disjoint*. To remove the disjoint nodes:

1. Select *Nodes / Select Disjoint Nodes*. You should get a message that two disjoint nodes were found and selected. Click *OK* to this prompt.
2. Select *Edit / Delete* or click the *Delete*  macro.

Now that the breakline has been inserted, triangular elements have been merged into quadrilaterals, and the disjoint nodes have been deleted, the mesh should look like that in Figure 3-19.

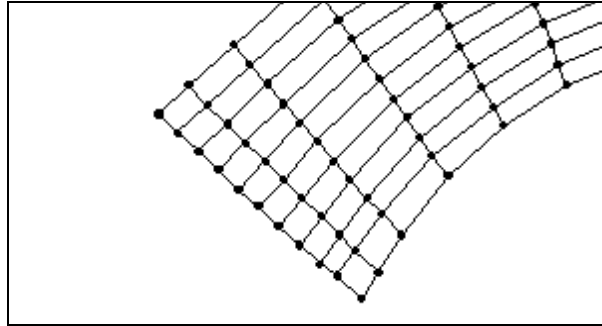

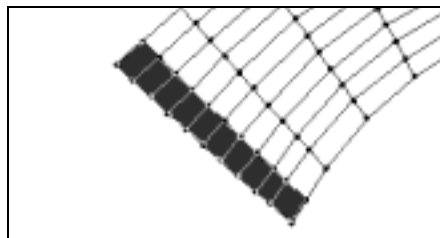


Figure 3-19 The final mesh after inserting the breakline.

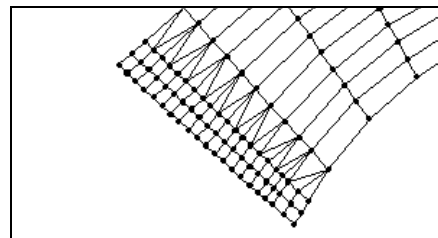
3.11.2 Using the Refine Command

Now that the breakline has been inserted, you are ready to use the refine command. This command splits a quadrilateral element into fourths. The reason the breakline was added is so that the refined elements would not be too skinny. You will refine the first column of elements on the very left side. To refine these elements:

1. Choose the *Select Elements*  tool from the *Toolbox*.
2. Hold the *CTRL* key and drag a line through the left-most column of elements, as shown in Figure 3-20a.
3. Select *Elements / Refine*. Each of the selected quadrilaterals will be split into four smaller quadrilaterals, and triangles will transition these small quadrilaterals to the larger quadrilaterals, as shown in Figure 3-20b.



(a). The elements to select.



(b). After the refine command.

Figure 3-20 The section of the mesh to refine.

3.12 Finishing the Mesh

Now that elements have been created and edited, the following things should be done before using this mesh in a finite element analysis:

- The Mesh Quality should be checked. You will see the same types of warnings as in section 3.10.
- The mesh should be renumbered. Remember that whenever nodes and elements are created, the mesh order should be fixed as in section 3.8.

3.13 Saving the Mesh

If *SMS* is registered, then the finite element mesh can be saved. This mesh will not be used in other tutorials, so saving it is not required. To save the mesh:

1. Select *File / Save As*.
2. Make sure the *Save as type* is set to *Project Files*.
3. Enter the name *poway1* and click the *Save* button.

3.14 Conclusion

This concludes the Mesh Editing tutorial. Although not every option was discussed, you should be familiar with many of the tools that *SMS* provides for mesh editing. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File / Exit*. If prompted to confirm, click the *Yes* button.

Basic RMA2 Analysis

4.1 Introduction

This lesson will teach you how to prepare a mesh for an *RMA2* simulation. You will be using the project file *stmaryout.spr* created in lesson 2. If you have not completed this lesson, the file needed for this tutorial can be found in the *tutorial\tut4* directory. If you ran *RMA2* in lesson 2, the project file will reference *stmaryout.sim*. (If you ran *FESWMS*, the project file will reference a *FESWMS* simulation file and you should open the project file *stmaryout.spr* in *tutorial\tut4*.) An *RMA2* .sim file is a type of super file. It contains a list of filenames that are used by *FESWMS*. The actual input data is stored in the files named in the super file. To open the file:

1. Select *File / Open*.
2. Open the file *stmaryout.spr*, either the one that you created in lesson 2 or the one from the *tutorial\tut4* directory.
3. If you still have geometry open from a previous tutorial, you will be warned that all existing data will be deleted. If this happens, click the *OK* button to the prompt.

The conceptual model automatically applied the flow and head boundary conditions when the mesh was generated.

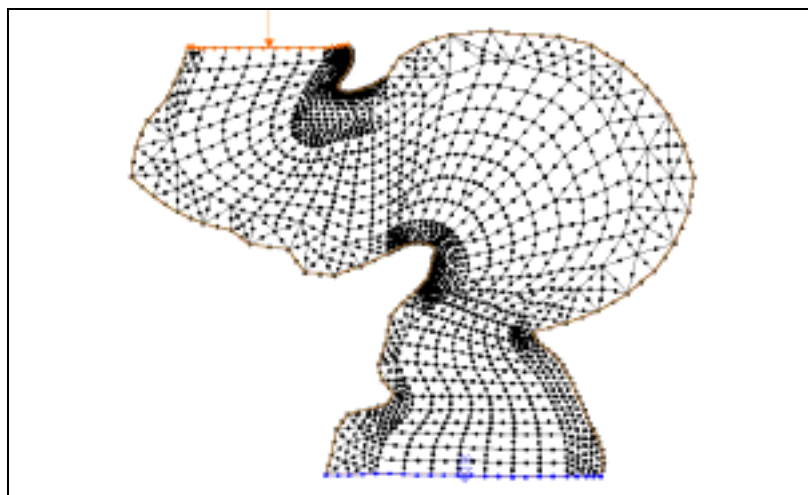


Figure 4-1. The mesh from *stmaryout.sim*.

4.2 Defining Material Properties

Each element of the mesh is assigned a material type ID. When the file was opened, the materials were created with default parameters. These material properties must be changed for this particular mesh. The material properties define how water flows through the element (see the *SMS Help* for details of what each parameter represents). To edit the material parameters:

1. Select *RMA2 / Material Properties*.
2. In the *RMA2 Material Properties* dialog, highlight the material *main_channel* (ID 1).
3. Make sure the *Isotropic eddy viscosities* option is turned on and enter a value of 25 for the eddy viscosity (E).
4. Set the Manning's roughness value (n) to 0.020.
5. Highlight the material *left_channel* (ID 2) and set E to 25 and n to 0.03.
6. For the material labeled *right_channel* (ID 3), assign an E of 30 and an n of 0.003.
7. Click the *Close* button to close the *RMA2 Material Properties* dialog.

The material properties have now been properly defined. Note: the material zones can be displayed by opening the *Display Options* dialog and turning on the *Materials* option under the 2D-Mesh tab.

4.3 Checking The Model

Before running an analysis, the model should be checked for completeness. SMS provides a model checker for each numerical model that it supports. Although passing the model checker will not guarantee that the model will converge, some of the more common mistakes will be reported. To run the model checker:

1. Select *RMA2 / Model Check*.
2. Click the *Run Check* button.

The *RMA2* model checker may report a warning that the mesh has not been renumbered during this session. In this case, the warning can be ignored because the model was already renumbered in a previous session.

4.4 Saving The Simulation


The flow and head boundary conditions were previously defined inside the map module. The entire simulation can now be saved. To save the simulation:

1. Select *File / Save as*.
2. Make sure the *Save as type* is *Project Files* and enter the name *stmary2.spr*.
3. Click the *Save* button to save the simulation.

After saving the simulation, the analysis can be run.

4.5 Using GFGEN

Before running the finite element analysis, the ASCII geometry file created by *SMS* must be converted to a binary format that *RMA2* can understand. This is done with a program called *GFGEN*. To launch the *GFGEN* program:

1. Select *RMA2 / Run GFGEN*.
2. If a message such as “gfgv435.exe – not found” is given, click the *File Browser* button  to manually find the *GFGEN* executable.
3. Click the *OK* button to launch *GFGEN*.

When the *GFGEN* window finishes, a beep may sound and you will get a message to press the *RETURN* key. If you do not get such a message and the window goes away, then *GFGEN* crashed and you may want to contact technical support with your file.

4.6 Using RMA2

After *GFGEN* is successfully completed, the finite element analysis can run. *RMA2* is the analysis program that computes 2D flow solutions at each node. For the mesh used in this tutorial, a steady state solution will be computed. To launch the *RMA2* program:

1. Select *RMA2 / Run RMA2*. As with *GFGEN*, the prompt shows the location of the most recent *RMA2* executable.
2. If necessary, find the executable, and click the *OK* button to launch *RMA2*.

For this steady state simulation, *RMA2* should finish in a couple of minutes. When the simulation is finished, a beep may sound and you will be prompted to press the *RETURN* key. The file *stmary2.sol* will contain the *RMA2* solution data.

4.7 Conclusion

This concludes the Basic *RMA2* Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File / Exit*. If prompted to confirm, click the *Yes* button.

Basic FESWMS Analysis

5.1 Introduction

This lesson will teach you how to prepare a mesh for a *FESWMS* simulation. You will be using the project file *stmaryout.spr* created in lesson 2. If you have not completed this lesson, the file needed for this tutorial can be found in the *tutorial\tut5* directory. If you ran *FESWMS* in lesson 2, the project file will reference *stmaryout.fil*. (If you ran *RMA2*, the project file will reference an *RMA2* simulation file and you should open the project file *stmaryout.spr* in *tutorial\tut5*.) A *FESWMS* *.fil* file is a type of super file. It contains a list of filenames that are used by *FESWMS*. The actual input data is stored in the files named in the super file. To open the file:

1. Select *File / Open*.
2. Open the file *stmaryout.spr*, either the one saved from lesson 2 or the one supplied in the *tutorial\tut5* directory. The display will refresh with the mesh as shown in Figure 5-1.

The conceptual model automatically applied the flow and head boundary conditions when the mesh was generated.

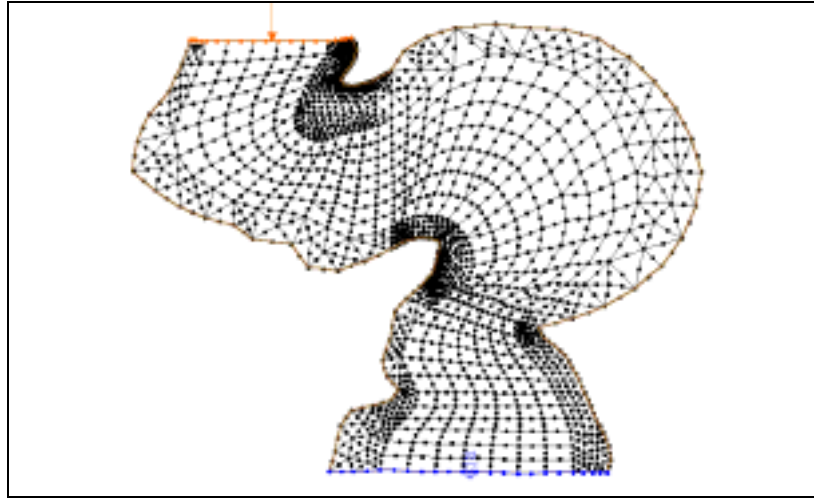



Figure 5-1. The mesh contained in *stmaryout.fil*.

5.2 Converting Elements

For *FESWMS*, it is best to use 9-noded quadrilateral elements (quads) even though both 8-noded and 9-noded quads are supported. The mesh contains 8-noded quads. To convert these to 9-noded quads:

- Select *Elements / QUAD8<->QUAD9*.

The screen will refresh and the quadrilateral elements will have 9 nodes. Since there was a change in the number of nodes, the mesh should be renumbered, even though it was renumbered before being saved. To do this:

1. Choose the *Select Nodestrings*  tool from the *Toolbox*.
2. Click in the icon at the downstream boundary condition.
3. Select *Nodestrings / Renumber* and click the *OK* button.

5.3 Defining Material Properties

Each element in the mesh is assigned a material type ID. After reading the file, *SMS* creates the materials with default values, which must be changed for this simulation. To change the material values:

1. Select *FESWMS | Material Properties*. In the upper right of the *FESWMS Material Properties* dialog, an image shows what the Manning's coefficients are for different depths.
2. Highlight the material named *main_channel* (ID 1) and enter the following values:
 - 0.025 for both *Manning coefficients (n1 and n2)*.
 - 20.0 for *Vo* and 0.6 for *Cu1* (near to bottom of the dialog).
3. Highlight the material labeled *left_channel* (ID 2) and enter the same values that were entered for the *main_channel* except enter 0.03 for Manning's coefficients.
4. Highlight the material labeled *right_channel* (ID 3) and enter the same values that were entered for the *left_channel*.
5. Click the *Close* button to close the *FESWMS Material Properties* dialog.

The eddy viscosity and Manning's roughness values should always be set. Other material properties can also be set for more advanced problems. See the *FESWMS* documentation for more information on these other material properties.

Optional: The materials can be displayed by opening the *Display Options* dialog and toggling the *Materials* option on. If you do this, be sure to turn the option back off before continuing with this lesson.

5.4 Setting Model Controls

Before running an analysis with this mesh, certain model controls and parameters must be set. The parameters and files used are specified in the *FESWMS Control* dialog. To change the global parameters:

1. First, to set the units to English, go to *Edit | Current Coordinates*.
2. Make sure the *Horizontal System* is *Local* and the *Horizontal* and *Vertical Units* are set to *U.S. Survey Feet*. Press *OK* to exit the units dialog.
3. Select *FESWMS | Model Control*.
4. In the *FLO2DH Input* section, turn on the *NET File* option and turn off all the other options.
5. In the *FESWMS Version* section, choose *FESWMS 3.**.
6. Make sure the *Solution Type* is set to *Steady state*.

7. Click the *Parameters* button and set the values shown below:

- Water-surface elevation = 20.0
- Unit flow convergence = 0.001
- Water depth convergence = 0.001
- Element drying / wetting = ON

Leave the other defaults and push *OK*.

8. Click the *Iterations* button and set the number of *Iterations* to 5. Then click the *OK* button to return to the *FESWMS Control* dialog.

9. Click the *Print* button and make sure the *ECHO to screen* option is turned on. Then click the *OK* button to return to the *FESWMS Control* dialog.

10. Click the *OK* button to exit the *FESWMS Control* dialog.

5.5 Model Check

Before running an analysis, the model should be checked for completeness. *SMS* provides a model checker for each numerical model that it supports. Although passing the model checker will not guarantee that the model will converge, some of the more common mistakes will be reported. To run the model checker:

1. Select *FESWMS / Model Check*.
2. Click the *Run Check* button.

The model checker should not report any warnings.

3. Click the *Done* button to exit the *FESWMS Model Checker*.

5.6 Saving The Simulation


Now that model control parameters and material properties have been defined, and the model has been checked, the simulation is ready to be saved. To do this:

1. Select *File/ Save As*.
2. Make sure the *Save as type* is set to *Project Files*, enter the name *stmary3*, and push *Save*.

The model control options and boundary conditions are saved to the file *stmary3.dat*, and the finite element network is saved to the file *stmary3.net*. If desired, look at the file *stmary3.fpr* to see these filenames.

5.7 Using FLO2DH

You are now ready to run the analysis. The analysis module of *FESWMS* is called *FLO2DH* and it can be launched from inside *SMS*. To launch the *FLO2DH* program:

1. Select *FESWMS / Run FLO2DH*.
2. If the prompt shows a message that *FLO2DH* is *not found*, click the *File Browser*  button manually find the correct program executable. (Be sure to use *flo2dh3x.exe*, the version 3 executable. A common error is to specify version 3 in model control and then to use *flo2dh.exe*, the version 2 executable).
3. Click the *OK* button to launch *FLO2DH*.

A new window will open in which *FLO2DH* will run the simulation. *FLO2DH* may take a few minutes to run, depending on the speed of your computer. When it is done, you will be prompted to press the *RETURN* key. If you are not prompted to do this, and the window goes away, then *FLO2DH* encountered an internal error and crashed. In this case, you may want to contact technical support with your files.

5.8 Conclusion

This concludes the Basic FESWMS Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File / Exit*. If prompted to confirm, click the *Yes* button.

Post Processing

6.1 Introduction

The solutions computed by the finite element analysis codes can be viewed in *SMS*. This is called *post-processing* of the finite element models. In this lesson, you will learn how to import, manipulate, and view solution data. You will need the geometry file *ld.geo* and the solution file *ld0dyn.sol* created by a dynamic *RMA2* simulation.

6.2 Data Sets

When *SMS* imports a solution file, it creates a *data set* for each quantity in the file. A data set contains solution data at each node of the mesh. *RMA2*, *FESWMS*, and *HIVEL2D* solutions create data sets representing velocity, water surface elevation, and water depth. *FESWMS* can create a special file with additional data sets for shear stress, Froude number, and energy head. An *RMA4* solution creates a data set for the concentration of each constituent. A *SED2D-WES* solution creates data sets representing sediment concentration and change in bed elevation. *ADCIRC* and *CGWAVE* solutions create data sets representing phase, angle, and amplitude of waves. *ADCIRC* also creates water surface elevation and velocity data sets. For steady state simulations, each data set only has a single time step, while in dynamic simulations, each data set has multiple time steps.

6.3 Using The Data Browser

All solution files are opened into *SMS* through the *File / Open* or through the *Data Browser* dialog. We will use the *Data Browser* in this tutorial. Solutions may only be imported after the corresponding mesh has been opened. Furthermore, if the mesh is edited so that it no longer corresponds to a solution file, *SMS* will not open the solution file, and it will close any solutions that have already been opened. To open the finite element network for this example:

1. Select *File / Open*.
2. Open the file *ld.geo* from the *tutorial\tut6* directory. A *.geo* file is an RMA2 mesh file.

With the geometry opened, the solution can be imported. To import the solution file:

1. Select *Data / Data Browser*.
2. Click the *Import* button.
3. Open the file *ld0dyn.sol*. This solution has 57 time steps in it. By default, the last time step of the velocity data set is highlighted.
4. Click the *Done* button to exit the *Data Browser* dialog.

When importing solution files, it is important to tell *SMS* which solution type is being opened. Table 6-1 shows which option to use for the different models.

Table 6-1 File formats for importing simulations.

File Type	Files Supported
Generic file	HIVEL2D, SMS-created data files
TABS file	RMA2, RMA4, SED2D-WES, RMA10
FESWMS file	FLOMOD, FLO2DH
ADCIRC file	ADCIRC hydrodynamics
ADCIRC harmonic file	ADCIRC wave information
CGWAVE file	CGWAVE

After closing the *Data Browser* dialog, the display will refresh. If contours or vectors are being displayed, the new display will correspond to the data from the solution file. See the *SMS Help* for information on changing the display of contours and vectors.

6.4 Creating New Data Sets With The Data Calculator

SMS has a powerful tool, called the *Data Calculator*, for computing new data sets by performing operations on scalar values and existing data sets. In this example, a data

set will be created, which contains the Froude number at each node. The Froude number is given by equation:

$$\text{Froude Number} = \frac{\text{Velocity Magnitude}}{\sqrt{\text{gravity} * \text{water depth}}}$$

To create the Froude number data set:



1. Select *Data / Data Calculator*.
2. Under the *Time Steps* section, turn on the *Use all time steps* option. This will compute the new function for each time of the dynamic simulation.
3. Highlight the *velocity mag* data set and click the *Add to Expression* button. The *Expression* will show “b:all”. The letter ‘b’ corresponds to the *velocity mag* data set and ‘all’ signifies all time steps.
4. Click the *divide* “ / “ button.
5. Click the *sqrt(x)* operation. Select the “??” text and delete it. This is just a placeholder to make sure you know that something should be placed there.
6. Type the gravity value for English units, 32.17405.
7. Click the *multiply* “ * “ button, then highlight the *water depth* data set and click the *Add to Expression* button.
8. Click the *closing parenthesis* “) “ button.
9. The expression should now read: “b:all / sqrt(32.17405 * c:all)”, where ‘b’ is the letter representing the velocity data set and ‘c’ is the letter representing the water depth data set.
10. In the *Result* field, enter the name *Froude* and then click the *Compute* button. *SMS* will take a few moments to perform the computations. When it is done, the *Froude* data set will appear in the *Data Sets* window.
11. Click the *Done* button to exit the *Data Calculator* dialog.

The display will refresh, showing contours of the Froude number data set. It can be treated just as any other dynamic scalar data set and can be saved in a generic data set file. See the *SMS Help* for more information on saving data sets.

6.5 Creating Animations

A film loop is an animation created by *SMS* to display changes in data sets through time. Flow trace and particle trace animations are a special type of film loop, which

use vector data sets to trace the path that particles of water will follow through the flow system. Only the visible portion of the mesh will be included in the film loop when it is created. For this tutorial, zoom into the portion of the mesh shown in Figure 6-1. To do this:

1. Be sure you are in plan view by clicking the *Plan View*  macro button.
2. Select the *Zoom*  tool from the *Toolbox*.
3. Click and drag the mouse to create a box about the area in Figure 6-1.

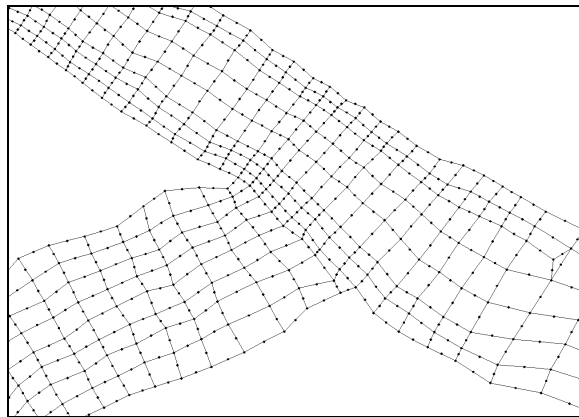




Figure 6-1. The area about which to zoom for the film loops.


6.5.1 Creating a Film Loop Animation

The following film loop will show how the velocity changes through time. To create and run the film loop:

1. Make sure that the *velocity mag* (scalar) and *velocity* (vector) data sets are active by selecting them from the *Scalar* and *Vector* drop down lists in the *Edit Window*.
2. Select *Data / Film Loop*.
3. Select *Scalar/Vector Animation* for the *Film Loop Type*.
4. Click the *File Browser*  button and enter a filename. Then click *Next*.
5. In the *Time Step Options* page, turn on both the *Scalar data set* and *Vector data set* options. All time steps should be selected, so click the *Next* button.
6. In the *Display Options* page, click the *Display Options*  button. Go to the *Mesh* tab and turn off all options except *Vectors*, *Contours*, and the *Mesh boundary* (do not click *OK* yet).


7. Click the *Vectors* tab and choose the *Define min and max length* option from the *Shaft Length* drop down list. Set the values for a *Min* of 10 and *Max* of 50. Also, choose the *Arrow placement: Display vectors on a grid* with a spacing of 30 in both directions.
8. Click the *OK* button to close both the *Display Options* dialog, still leaving the *Film Loop Options* dialog open.
9. Click the *Finish* button in the *Film Loop Setup* wizard to create the film loop.


SMS will display each frame of the film loop as it is being created, and a prompt in the *Edit Window* shows the frame being created. When the film loop has been fully generated, it will launch in a new *Play AVI Application* (PAVIA) window. This application contains the following controls:


Play  *button*. This starts the playback animation. During the animation, the speed and play mode can be changed.


Speed. This increases or decreases the playback speed. The speed depends on your computer.

Stop  *button*. This stops the playback animation.

Step  *button*. This allows you to manually step to the next frame. It only works when the animation is stopped.

Loop  *play mode*. This play mode restarts the animation when the end of the film loop has been reached.

Back/forth  *play mode*. This play mode shows the film loop in reverse order when the end of the film loop has been reached.


Copy  *button*. When the animation is stopped, this copies the current frame to the clipboard so it can be pasted into another application.

The generated film loop will be saved in the AVI file format. AVI files can be used in software presentation packages, such as Microsoft PowerPoint or WordPerfect Presentations. A saved film loop may be opened from inside SMS or directly from inside the PAVIA application (pavia.exe is located in the SMS installation directory and can be freely distributed).

6.5.2 Creating a Flow Trace Animation

A flow trace animation can be created if a vector data set has been opened. The flow trace randomly introduces particles into the network and follows each through the vector field. Steady state vector fields can be used in a flow trace animation to show flow direction trends. For dynamic vector fields, the flow trace animation can trace a single time step, or it can trace the changing flow field.

Note that a flow trace takes longer to generate than the scalar/vector animation. The bigger the window, the larger the animation and the more memory is required to generate it. If you have problems with this operation, decrease the size of the SMS window and try again. To create and run a flow trace film loop:

1. Once again, *Select Data / Film Loop*.
2. In the *upper left* of the *Film Loop Setup* wizard, select the *Flow Trace* option and change  the file name. Then click the *Next* button.
3. Make sure all the time steps of the velocity vector data set selected and click the *Next* button.
4. Leave all the options in the *Flow Trace Options* page as their default values and click the *Next* button.
5. Click the *Finish* button to generate the animation.

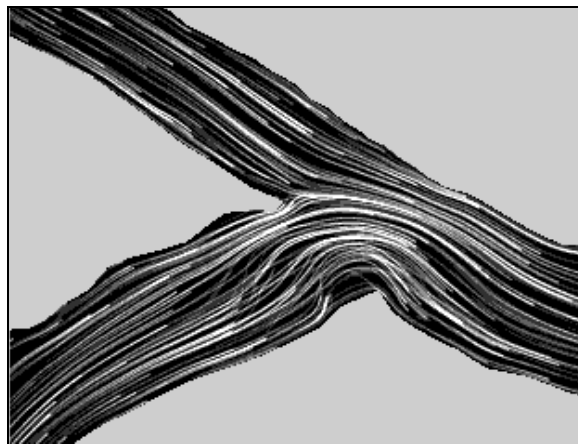






Figure 6-2. One frame from the Id1 flow trace.

After a few moments, the first frame of the flow trace animation will appear on the screen. As before, the frames are generated one at a time, and a prompt shows which frame is being created. When the flow trace has been created, it is launched in a new window, just as the previous animation.

- View the flow trace using the same controls as with the film loop animation.

6.6 Drogue Plot Animation

Drogué plot animations are similar to flow trace animations, except that they allow the user to specify where particles will start. A particle/drogué coverage defines the starting location for each particle. To create this coverage:

1. Switch to the *Map*  module.
2. Select *Feature Objects / Coverages* and change the active coverage type to *Particle/Drogué*. Then click *OK*.
3. Create two feature arcs with the *Create Feature Arcs*  tool, one across each upstream branch of the river.
4. Select both arcs with the *Select Feature Arcs*  tool by holding *SHIFT* while clicking on each, and choose *Feature Objects / Redistribute Vertices*. Specify the *Number of Segments* to be 20 and click *OK*.
5. Create three individual points in the downstream branch of the river with the *Create Points*  tool.

The arcs and points that were defined should look something like Figure 6-3. For the drogué plot animation, one particle will be created at each feature point each feature arc vertex.

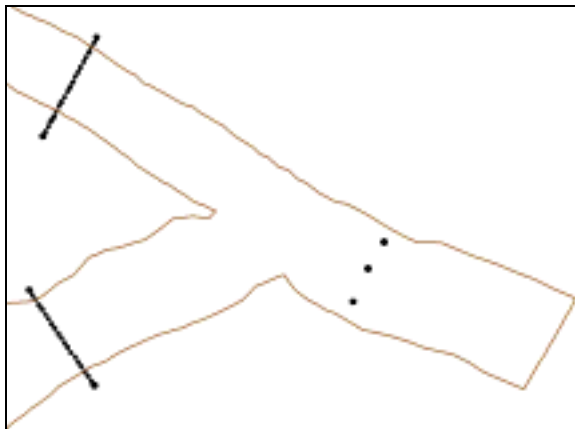




Figure 6-3. Feature objects for the particle/drogué coverage.

To create the drogué plot animation:

1. Switch back to the *Mesh*  module and select *Data / Film Loop*.
2. Select the *Drogué Plot* animation type and change  the filename, then click the *Next* button (the coverage was just created so it is already set).

3. In the Time Step section, set the Start Time to 0.0 and the End Time to 5.0. Set the number of timesteps to 50. Click the *Next* button.
4. In the *Color Options* section, associate the color ramp with the *Distance traveled*, and set the *Maximum distance* to 1500. Turn on the *Write report* option, then click *Next*.
5. Click the *Finish* button to generate the animation.

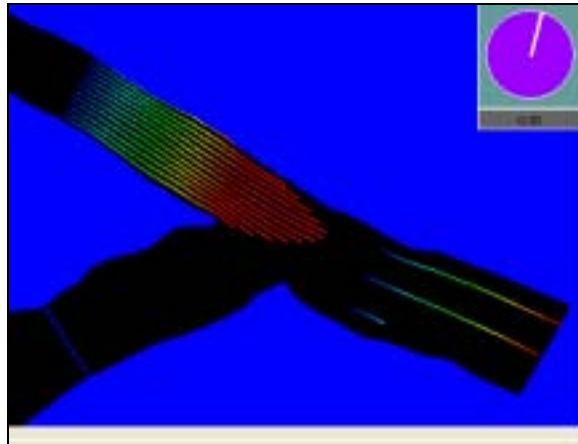


Figure 6-4. Sample drogue plot animation.

The drogue plot animation generates a report by turning on the option in step number 4 above. To see this report:

- Choose *File / View Data File* and Open the file “drogues.pdr”.

Now that you have seen the three main animation types available in SMS 8.0, feel free to experiment with some of the available options, especially with the flow trace and drogue plot animation types.

6.7 2D Plots

Plots can be created to help visualize the data. Plots are created using the observation coverage in the map module. There is a special tutorial on the observation coverage, which shows how to create data plots in lesson 17.

6.8 Conclusion

This concludes the Post Processing tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File / Exit*. If prompted to confirm, click the *Yes* button.

Advanced RMA2 Analysis

7.1 Introduction

This lesson will teach you how to use revision cards for a spin down simulation in *RMA2*. The geometry has already been created and renumbered. To open the file:

1. Select *File / Open*.
2. Open the file *ld.geo* from the *tutorial\tut7* directory. If you have mesh data open from a previous tutorial, you will be warned that all existing data will be deleted. If this happens, click the *OK* button to the prompt.

The geometry data will open, as shown in 7-1.

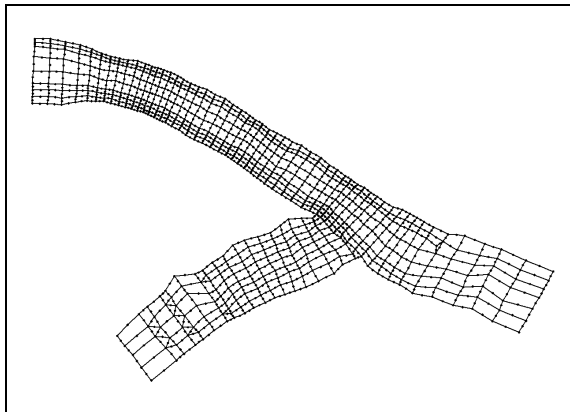


Figure 7-1. The mesh contained in the file *ld.geo*.

7.2 Defining Material Properties


Each element in the mesh is assigned a material type ID. The materials were created with default parameters that must be changed for this particular simulation. The material properties define how water flows through the element (see the *SMS Help* for more information). To edit the material parameters:

1. Select *RMA2 / Material Properties*.
2. In the *RMA2 Materials Properties* dialog, highlight *material 01*.
3. Make sure the *Isotropic eddy viscosities* option is on and enter a value of 25 for the eddy viscosity (*E*).
4. Enter 0.03 for Manning's roughness (*n*).
5. Highlight *material 02* and set *E* to 50 and *n* to 0.04.
6. Click the *Close* button to close the *RMA2 Material Properties* dialog.

The material parameters have now been defined. You may turn on the display of the materials if you want. To do this, open the *Display Options* dialog and turn on the *Materials* option under the 2D-Mesh tab. If you do this, turn them back off before continuing.

7.3 Creating Nodestrings

For this tutorial, flow and water surface elevation will be defined along nodestrings at the *open boundaries* of the mesh. An open boundary is a boundary where flow enters or exits from the mesh. Generally for *RMA2*, flow is specified at the inflow or upstream boundaries and water surface elevation (head) is specified at the outflow or downstream boundaries. Three boundary strings must be created, one at each of the open boundaries. These boundaries are highlighted in Figure 7-2. To create a nodestring across the upper left boundary:

1. Choose the *Create Nodestrings* tool  from the *Toolbox*.
2. Start the nodestring by clicking on the upper left corner node.
3. Hold the *SHIFT* key and click on the bottom node of the inflow boundary. *SMS* will automatically select all nodes between the two.
4. Press the *ENTER* key to terminate the nodestring.

In a similar fashion, create nodestrings at the other two open boundaries.

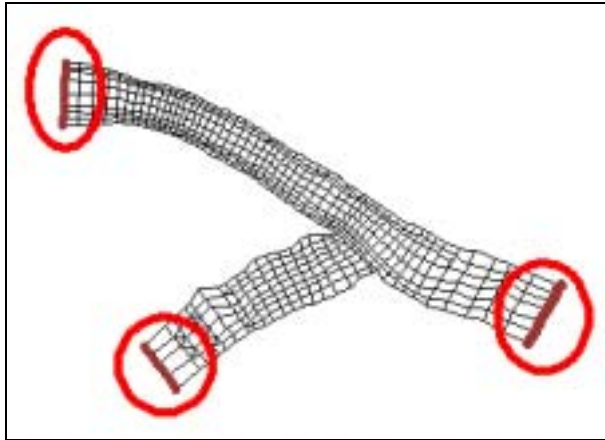


Figure 7-2. Position of the boundary nodestrings in the mesh.

7.4 Defining a Dynamic Simulation

The finite element network and associated material properties are only part of the numerical model. In addition, several other model control parameters must be defined. These model parameters include items such as how to handle wetting and drying, the model units, simulation time, and how many iterations should be made to get the solution to converge. Information on each parameter can be found in the *SMS Help* and the *RMA2* documentation.

Before continuing, make sure that the units are English. To do this:

1. Select *Edit | Current Coordinates*.
2. Make sure the *Horizontal System* is set to *Local* and the *Horizontal* and *Vertical Units* are set to *U.S. Survey Feet*.
3. Click *OK* to exit the dialog.

For this simulation, a very low water surface will be defined. However, to be able to assign the water surface value, a hotstart file will be generated. The desired head value is so low that a few elements may become dry, but the marsh porosity option will be used. To define the model parameters:

- Select *RMA2 | Model Control*.

This opens the *RMA2 Model Control* dialog in which various parameters will be set in the next few sections.

7.4.1 Defining Time Parameters

The first simulation should be a dynamic simulation with 57 time steps. To do this:

1. Change the *Solution Type* (bottom right) to *Dynamic*.
2. In the *Computation Time* section, use the *Specify time step time* option with a *Time step* of 10.0 (hours), and then 57 for the *Number of time steps*.

7.4.2 Defining Hotstart Output

This simulation will be used to create a hotstart file for a steady state simulation. To do this:

- Turn on the *Hotstart output file* option (middle left).

7.4.3 Defining Iteration Controls

Since the water surface for this model will become rather low, and thus unstable, the number of iterations should be increased and a convergence parameter should be defined to make sure the model converges. To do this:

1. Set both the *Initial solution* and *Each time step* iterations to 20.
2. Turn on both the *Steady state depth convergence* and the *Dynamic depth convergence* options and enter a value of 0.001 for each.

Each time step of the model will be considered converged when the maximum change in water depth at all nodes is less than 0.001 ft.

7.4.4 Defining Wetting/Drying Parameters

Wetting/drying parameters are not required, so they are in the *Optional BC Controls* dialog. To set these parameters:

1. Click the *Optional BC Controls* button.
2. Turn on the *Dry Elements* (top right) option.
3. Set the *Check wet/dry testing* iterations to 4, turn on and set the *Nodal dry depth* to 0.001 ft, and turn on and set the *Nodal active depth* to 0.10 ft.
4. Turn on the *Marsh Porosity* option and click the *Options* button. Use the *Distance below each node's bathymetry value* as 4.0. Set the *Transition Range* to 1.0 and the *Minimum wetted surface area* to 0.01. Then click *OK*.

To accept all the above values:

- Click the *OK* button to close both model control dialogs.


7.5 Defining Transient Boundary Conditions

There are various uses for transient boundary conditions, such as to model a tide (transient head) or run a storm hydrograph through a river (transient flow). This example uses transient boundary conditions to create a hotstart file for increasing the model stability.

The water surface elevation will be spun down to a desired starting condition so that a steady state model can be run, while the flow conditions will be constant. The head curve will be imported from a file. (See the *SMS Help* for information on creating time series curves.)

7.5.1 Defining Flow Boundary Conditions

To assign the flow conditions:

1. Choose the *Select Nodestrings*  tool from the *Toolbox*. An icon appears at the center of each nodestring.
2. Select the top left nodestring by clicking in its icon.
3. Select *RMA2 / Assign BC*.
4. In the *RMA2 Assign Boundary Conditions* dialog, set the *Condition Type* to *Flow BC* and assign a *Flowrate* of 55,000 (cfs).
5. Make sure the *Flow direction* is set to *Perpendicular to Boundary*.
6. Click the *OK* button to exit the *RMA2 Assign Boundary Conditions* dialog.

This defines the left nodestring to be an inflow boundary condition. To define the second inflow boundary condition:

1. Select the bottom left nodestring.
2. Assign a perpendicular flow of 580 cfs.

7.5.2 Defining Head Boundary Conditions

The head boundary condition will be a transient curve and will be imported from a file. To assign this boundary condition:

1. Select the nodestring on the right boundary.
2. Select *RMA2 / Assign BC*. Make sure the *Head BC* option is selected.

3. Change the boundary condition type to *Transient* and then click the *Define curve* button.
4. In the *XY Series Editor* dialog, click the *Import* button and open the file *head_ld.xys*. A transient curve will be displayed in the window.
5. Click the *OK* button in both dialogs to assign the boundary condition.

7.6 Running The Model Checker

Before running the analysis, it is a good idea to run the model checker. The model checker looks for potential problems and common mistakes in the mesh. To run the model checker:

1. Select *RMA2 / Model Check*.
2. In the *RMA2 Model Checker* dialog, click the *Run Check* button.

The model checker may come up with a warning. This warning says that the mesh has not been renumbered since it was opened. This can be ignored because, as stated earlier, the mesh was already renumbered in an earlier session. Although the model checker does not come up with any warnings, this does not mean that the model will converge. However, it means that there should not be any serious errors. Although the *RMA2 Model Checker* dialog can be open while using *SMS*, it will not be needed anymore. To close it:

- Click the *Done* button to close the *RMA2 Model Checker* dialog.

7.7 Saving The Simulation

Now that the boundary conditions and global parameters have been defined and checked, the simulation can be saved. The *TABS* programs have incorporated a way to be launched from inside *SMS* after a simulation is saved. To save the simulation:

1. Select *File / Save As*.
2. Make sure the *Save as type* is set to *Project Files* and enter *ld0.spr* as the filename.
3. Click the *Save* button to save the project.

7.8 Using Revision Records

The simulation that has been saved is a dynamic simulation that will produce a hotstart file. This hotstart file will be used as input to a steady state model. However, a dynamic hotstart file cannot be used as input into a steady state model. Because of this, a change needs to be made to the boundary condition file to convert the time steps into revisions. Using revisions causes *RMA2* to see a steady state model, which can be used to hotstart a new steady state model. To make this change to the boundary condition file:

1. Select *File / View Data File*.
2. Select *ld0.bc* and push *Open*.
3. If you want to use a text editor different from the default, enter the text editor name and click the *OK* button.
4. Change each *END* record to *REV*, except for the last one. The file will look like this:

```
REV  Simulation at time =  0.00
.
.
REV  Simulation at time = 10.00
.
.
END  Simulation at time = 570.00
STOP
```

NOTE: You may want to use the *Search / Replace* feature if you are using the default *Notepad* on the pc.

5. Save the file and close the text editor.


The simulation can now be run through *RMA2*. Do not save the project again because the *REV* cards will again be replaced by *END* cards.

7.9 Running The Initial Simulation

7.9.1 Running GFGEN

Before running the finite element analysis, the ASCII geometry file must be converted to a binary file that *RMA2* can understand. This is done with a program called *GFGEN*. To launch the *GFGEN* program:

1. Select *RMA2 / Run GFGEN*. A prompt appears, showing the location of the most recent *GFGEN* executable.

2. If a message such as “gfgv435.exe – not found” is given, then click the *File Browser*  button to manually find the *GFGEN* executable.
3. Click the *OK* button to launch *GFGEN*.

When the *GFGEN* window finishes, you will get a message to press the *RETURN* key. If you do not get such a message and the window goes away, then *GFGEN* crashed and you may want to contact technical support with your geometry file.

7.9.2 Running RMA2


After *GFGEN* is successfully completed, the finite element analysis can run. To launch the *RMA2* program:

1. Select *RMA2 / Run RMA2*. As with *GFGEN*, the prompt shows the location of the most recent *RMA2* executable.
2. If necessary, find the executable, and click the *OK* button to launch *RMA2*.

Once again, a window will appear, this time running *RMA2*. After the model runs, you should be prompted to press the *RETURN* key. When the solution has been finished, a file named *ld0.hot* will be created, which can be used as an initial condition file for a steady state simulation.


7.10 Defining a Steady State Simulation

Now that the hotstart file has been generated, the steady state simulation can be created. Most of the model parameters will remain the same as those from the “dynamic” simulation, such as wetting/drying and model units. There are only a few changes that need to be made. To make these changes:

1. Select *RMA2 / Model Control*.
2. Change the *Solution Type* to *Steady State*. Notice that the *Time Control* and *Dynamic Depth Convergence* sections become unavailable.
3. Turn on the *Hotstart input file* option and click on the  button.
4. Select *ld0.hot* and push *Open*. This will make *RMA2* look for a hotstart file to use as the initial conditions.
5. Click the *OK* button to close the *RMA2 Model Control* dialog.

7.11 Defining Constant Boundary Conditions

Constant boundary conditions, as the name implies, do not change with time. The inflow boundaries were initially defined as constant boundary conditions. For the steady state simulation, these will remain the same. The head boundary needs to be changed to a constant condition. To do this:

1. Choose the *Select Nodestrings*  tool from the *Toolbox*.
2. Select the right nodestring (head boundary) by clicking in its icon.
3. Select *RMA2 / Assign BC*.
4. In the *RMA2 Assign Boundary Conditions* dialog, the boundary condition is automatically changed to Constant. Enter a value of 40.0 (feet).
5. Click the *OK* button to close the *RMA2 Assign Boundary Conditions* dialog.

The definition of the steady state simulation is now complete.

7.12 Saving The New Simulation

This simulation needs to be saved. To do this:

1. Select *File / Save As*.
2. Make sure the *Save as type* is set to *Project Files* and enter the name *ld1.spr*.
3. Push the *Save* button.

This saves the new steady state simulation so that *RMA2* can be run.

7.13 Running The Final Simulation

After saving this simulation, *RMA2* can be run to generate the final solution. To launch the *RMA2* program:

1. Select *RMA2 / Run GFGEN*.

NOTE: If you had saved the simulation with the same name, *ld0*, you would not have needed to run *GFGEN* again. *GFGEN* only needs to be run when the mesh geometry changes or when you save with a new filename.

2. After *GFGEN* is finished running and you have pushed *Enter*, select *RMA2 / Run RMA2*.

3. Click the *OK* button to the prompt.

Once again, a window will open while *RMA2* runs. The file *ld1.sol* is the final solution file, and contains the steady state flow and head data at each node.

7.14 Conclusion

This concludes the Advanced *RMA2* Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File / Exit*. If prompted to confirm, click the *Yes* button.

Advanced FESWMS Analysis

8.1 Introduction

This lesson will teach you how to prepare an advanced *FLO2DH* simulation, including the use of weirs. You will be using the file *suecreek.spr* as shown in Figure 8-1. This mesh has been created and renumbered in a previous session. Although *FLO2DH* supports both 8- and 9- noded quadrilateral elements, this mesh contains only 9-noded quadrilaterals. To open the mesh data:

1. Select *File / Open*.
2. Highlight the file *suecreek.spr* in the *tutorial* directory and click the *Open* button. If a mesh is already open, you will be warned that the previous mesh will be deleted. If this happens, click the *OK* button at the prompt.

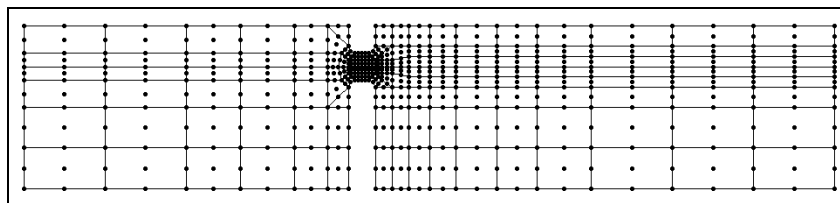


Figure 8-1. The *suecreek.fil* geometry.

8.2 Defining Material Properties

Each element of the mesh is assigned a material ID. The material ID tells *FLO2DH* which material properties should be assigned to the element. There are four different material types in this mesh, but the material properties have not been defined. When *SMS* opens a mesh with undefined materials, the materials are assigned default properties. See the *FESWMS* documentation for a definition of individual material parameters. To change the material values:

1. Select *FESWMS / Material Properties*. A graphical image in the upper right corner of the *FESWMS Material Properties* dialog shows the Manning's n value as a function of water depth.
2. Highlight *material_01*, and be sure that its *ID* is 1.
3. Enter the following values:
 - 0.035 for both $n1$ and $n2$
 - 20.0 for Vo
 - 0.6 for $Cu1$
4. Select *material_02* and enter the same values that were entered for *material_01*.
5. For *material_03* and *material_04*, enter the same values as above, except that $n1$ and $n2$ are both 0.055.
6. Click the *Close* button to accept these changes and close the *FESWMS Material Properties* dialog.

You have just assigned values for the four materials in this mesh. Notice that there are only two distinct material regions because materials 1 and 2 have the same values, as do materials 3 and 4.

Optional: The materials can be displayed by opening the *Mesh Display Options* dialog and turning on the *Materials* option. You may turn on the display of materials, but if you do so, be sure to turn them off before continuing with this tutorial.

8.3 Creating The Hotstart File


We wish to model this portion of river with 9,000 cfs of flow. However, this model will not converge when using such a large initial flow rate. An intermediate solution file using a lower flow rate will first be created. This intermediate solution will then be used as a hotstart file so that a solution with the desired flow can be computed.

8.3.1 Assigning Boundary Conditions

Boundary conditions such as flow and head define how water enters and leaves the finite element network. Without proper boundary conditions, instability of the model and inaccuracy of the solution will result.

8.3.1.1 Defining Flow and Head

A steady state model such as this can only have constant boundary conditions. The flow and head boundary conditions will be defined at nodestrings on opposite sides of the model as shown in Figure 8-2. To create the two boundary nodestrings:

1. Choose the *Create Nodestrings*  tool from the *Toolbox*.
2. Click on the upper node of the left boundary.
3. Hold the *SHIFT* key and click on the lower node of the left boundary. This creates the nodestring all the way across the left boundary of the mesh. If you did not hold the *SHIFT* key, it would not be a valid boundary nodestring because it would not include all nodes across the boundary.
4. End the nodestring by pressing the *ENTER* key.
5. Repeat this procedure to create a second nodestring on the right boundary.

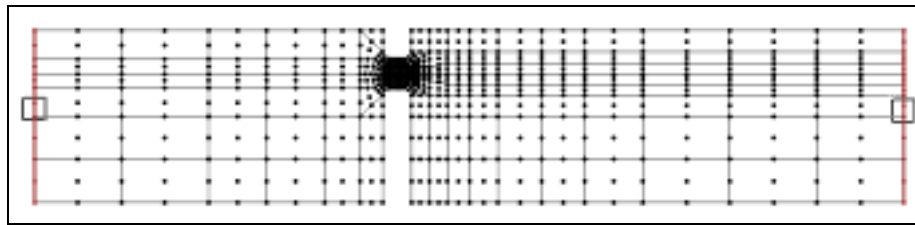



Figure 8-2. Location of the boundary condition nodestrings.

Boundary conditions can now be assigned to the nodestrings. To assign the flow to the left boundary:

1. Choose the *Select Nodestrings*  tool from the *Toolbox*. An icon appears at the center of each nodestring, as shown in Figure 8-2.
2. Select the nodestring on the left boundary by clicking inside its icon.
3. Select *FESWMS / Assign BC*.
4. In the *FESWMS Nodestring Boundary Conditions* dialog, turn on the *Flow* option and assign a value of 5000 (cfs). Be sure that the *Normal* option is selected.
5. Click the *OK* button to close the dialog.

The selected nodestring is now defined as a flow nodestring and its color changes. An arrow appears at the center of the nodestring to indicate the flow direction and the flow value is shown next to the arrow (see Figure 8-3).

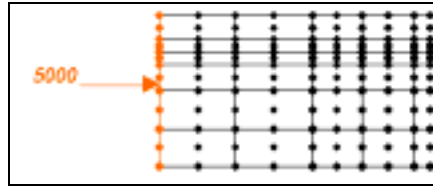


Figure 8-3 The inflow boundary condition.

To assign the head to the right boundary:

1. Select the right nodestring.
2. Select *FESWMS / Assign BC*.
3. In the *FESWMS Nodestring Boundary Conditions* dialog, turn on the *Water surface elevation* option and assign a value of 812.9 (feet). Be sure that the *Essential* option is selected.
4. Click the *OK* button to close the dialog.

The selected nodestring is now defined as a head nodestring and its color changes. A head symbol appears at the center of the nodestring and the head value is shown next to the symbol (see Figure 8-4).

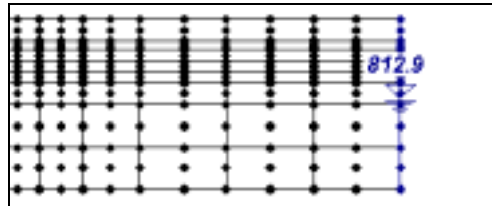


Figure 8-4 The outflow boundary condition.

8.3.2 Creating Weirs

With *FESWMS*, flow control structures such as weirs, piers, culverts, and drop inlets are easily added to the mesh. Weirs, culverts, and drop inlets are created between pairs of nodes. Wide structures can be created between strings of node pairs. For this model, a weir will be defined along five node pairs across the abutment at the bottom middle of the mesh, as shown in Figure 8-5.

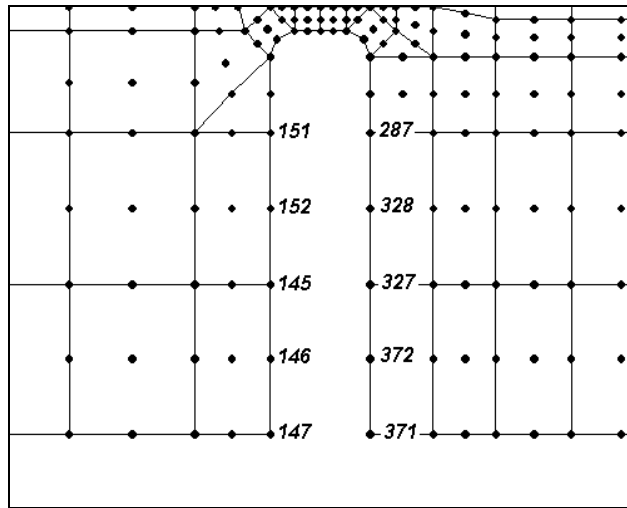




Figure 8-5 Area where weirs will be added.

Turn on the display of node numbers using the *Display Options*  dialog. Zoom in on the lower portion of the middle of the bridge as shown in Figure 8-5. The five node pairs for the weir are: 151<->287, 152<->328, 145<->327, 146<->372, and 147<->371. To select the first pair of nodes:

1. Choose the *Select Nodes*  tool from the *Toolbox*.
2. While holding the *SHIFT* key, click on nodes 151 and 287.

To create a weir segment between the selected nodes:

1. Select *FESWMS / Weir*.
2. Make sure the *upstream* node is 151, and the *downstream* node 287. If these are opposite, click the *Switch* button.
3. Enter the following values:
 - 0.53 for the *Cw - discharge*.
 - 25 for *Lw - Crest Length*.
 - 825 for the *Zc - Crest Elevation*.
4. Click the *OK* button or press the *ENTER* key.

The first weir section has now been defined. The last four weir sections are defined in a similar manner. To finish the weir:


- Select each pair of nodes and assign weir values as shown in Table 8-1.

Table 8-1 The weir node pair data.

Upstream	Downstream	Discharge	Length	Elevation
151	287	0.53	25	825
152	328	0.53	100	825
145	327	0.53	50	825
146	372	0.53	100	825
147	371	0.53	25	825

When these values are all entered, you will have defined a single weir that spans the two bottom elements. This weir is 300 feet long, has an elevation of 825 feet, and has a discharge coefficient of 0.53. There is a specific formula for determining the crest length of each pair of nodes. Each midside node has $2/3$ of the element width in its crest length while each corner node has $1/6$ of each element width that is involved in the weir. In this case, there are two elements involved in the weir, both of which are 150 feet long. This yields the distribution given in the table. See the *SMS Help* for more information on weirs and other flow control structures.

After creating the weir, reset the display to the way it was before starting the weir creation, as shown in Figure 8-6. To do this:

1. Turn off the display of node numbers using the *Display Options* dialog.
2. Click the *Frame*  macro in the *Toolbox* to frame the image.

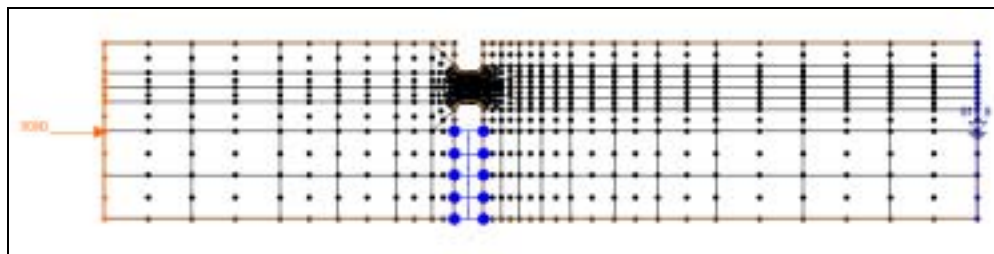


Figure 8-6. The final bridge geometry.

8.3.3 Saving The Data

With *FESWMS* software, data is saved in multiple files. The file names are specified in the *FESWMS Model Control* dialog. To set up the *FESWMS* file options:

1. Select *FESWMS / Model Control*.
2. In the *FESWMS Version* area, choose *FESWMS version 3.**.
3. In the *FLO2DH Input* section, turn off all options except for the *NET File*.


4. Click the Parameters button and make sure Element drying / wetting is on and push *OK*.
5. Click the *OK* button.

Now that these model control options have been set, the data is ready to be saved. To save the *FESWMS* data:

1. Select *File / Save As*.
2. Make sure the *Save as type* is *Project Files* and enter the name *suecreek2.spr*.
3. Click the *Save* button.

8.3.4 Using FLO2DH

You are now ready to run an analysis. The analysis module of *FESWMS* is called *FLO2DH*. To run *FLO2DH*:

1. Select *FESWMS / Run FLO2DH*.
2. If the prompt shows a message that *flo2dh* is *not found*, then click the *File Browser*  button and find the location manually.
3. Click the *OK* button to run *FLO2DH*.

A window will open to run the *suecreek* model through *FLO2DH*. Depending on the speed of your computer, *FLO2DH* will take a few minutes to finish the solution. Upon completion, *FLO2DH* writes a solution file named *suecreek2.flo*. This file contains the velocity and water surface elevation for each node in the mesh. The solution can be read into *SMS* using the *Data Browser* (see the *SMS Help* for more information).


Note: If you have not registered the *FESWMS* interface, you can still run the model in a prompt by using the supplied *suecreek2.** files in the *output* directory.

8.4 Reworking The Solution

As previously stated, the solution that was computed earlier is only an intermediate solution. It will be used as a hotstart file for an altered set of boundary conditions that will be set up in this section.


8.4.1 Changing The Boundary Conditions

The flow value needs to be increased to the full desired value. To do this:

1. Choose the *Select Nodestrings*  tool from the *Toolbox*.
2. Select the flow nodestring (left boundary) by clicking in the icon.
3. Select *FESWMS / Assign BC*.
4. Increase the flow value to 9000 (cfs).
5. Click the *OK* button

8.4.2 Editing The Weir Data


If you look at the previous solution data, you will see that there is no flow through the weirs. The water surface at the weirs is much less than the weir crest elevation, so overtopping did not occur. This was purposely done to add model stability. Now, however, the weirs need to be lowered to their true elevation of 812.5 ft. To do this:

1. Choose the *Select Nodes*  tool from the *Toolbox*.
2. Select one pair of nodes (select the nodes while holding the SHIFT key down) that makes up one of the weir sections. Each weir pair is shown connected by a line.
3. Select *FESWMS / Weir*.
4. Change the *Crest Elevation* to 812.5 feet.

Repeat these steps to lower the crest elevation of all weir segments to 812.5 feet.

8.4.3 Using The Hot Start file

SMS needs to tell *FLO2DH* to use the previous solution file as an input *hotstart*, or *initial condition*, file. To do this:

1. Select *FESWMS / Model Control*.
2. In the *FESWMS Control* dialog, turn on the *INI file* option.
3. Click the *File Browser*  button to the right of this option. Find and select the file *suecreek2.flo* that was created when *FLO2DH* ran. If you were not able to run the model, you can use the solution file that is provided in the *output* directory.

4. Click the *OK* button in both dialogs.

8.4.4 Computing a New Solution File Using a Hotstart File

To run the new simulation:

1. Using the steps described earlier, save the modified simulation file as *suecreek3.spr*. Be sure to save the new simulation in the same directory as the *suecreek2.flo* file or FLO2DH will not find *suecreek2.flo*.
2. Use the steps described earlier to run a new simulation. A new solution file named *suecreek3.flo* will be created by *FLO2DH*.

The solution file can be imported into *SMS* through the *Data Browser* (see the *SMS Help* for more information). Once a solution file has been read, *SMS* processes the data sets created from them in a model independent fashion. Therefore, the methods for post-processing described in Lesson 6 are applicable for *FESWMS* solution files.

8.5 Conclusion

This concludes the *Advanced FESWMS Analysis* tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File / Exit*. If prompted to confirm, click the *Yes* button.

SED2D-WES Analysis

This lesson will teach you how to use *RMA2* in conjunction with *SED2D-WES* to perform sediment transport simulations. The files required for this tutorial are 's.sim', 's.bc', 's.geo', 's_gf.run', 's_rm.run', and 'hydrograph.xys'. The *RMA2* simulation will model a 5,000 cms hydrograph through a simple 'S' channel bend with a base flow of 3,000 cms.

SED2D-WES requires the hydrodynamics to be computed elsewhere and given as part of its input. This is usually done with another *TABS* components, *RMA2*. An underlying assumption of *SED2D-WES* is that the bed does not change enough to significantly alter flow velocities. When large changes in the bed occur due to either erosion or deposition, a new hydrodynamic solution should be computed before continuing the *SED2D-WES* simulation.

Currently, there is no support in *SED2D-WES* for US units. Metric units should be used for all data, including the geometry and hydrodynamic simulation. Although it is possible to define conversion factors from US units for the geometry and the hydrodynamics, the SMS developers do not recommend such practice.

9.1 TABS Data Flow

In order to be successful at modeling with *SED2D-WES*, you need to understand the data flow through the *TABS* models. There are many files that are used for this type of modeling situation and it is easy to get lost in all the data. Figure 9-1 shows the most important files that are involved in the *SED2D-WES* analysis process. The files in this image are described below.

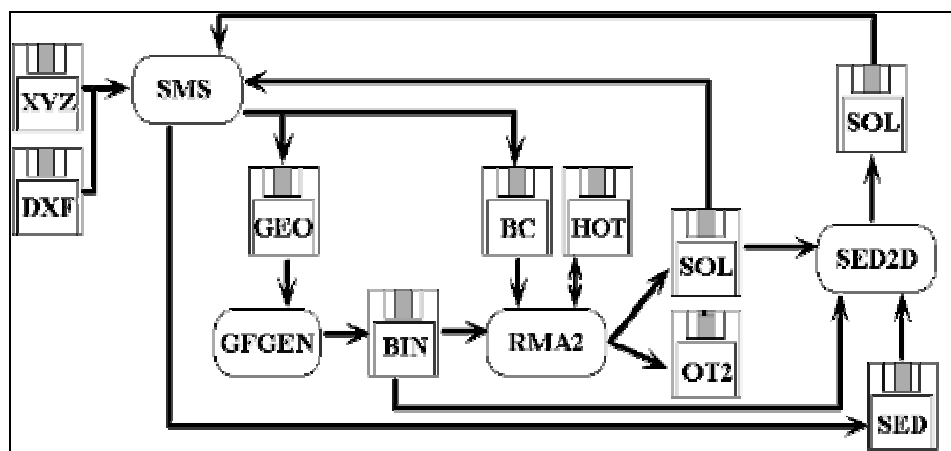


Figure 9-1. The TABS models data flow.

SMS saves three types of data that are required to perform a *SED2D-WES* simulation. These are the geometry (.geo) file, hydrodynamic boundary conditions (.bc) file, and sediment transport definition (.sed) file. The geometry is created by combining location and bathymetric data from a variety of sources such as surveys, dxf files, and USGS quad sheets. After the geometry is built, appropriate boundary conditions are applied to define the hydrodynamic and sediment simulations.

9.1.1 GFGEN

The geometry file saved by *SMS* is written in ASCII format. The *TABS* component *GFGEN* converts the ASCII geometry file into a binary geometry file (.bin). This binary geometry file is used by all other *TABS* modeling components to define the model domain.

9.1.2 RMA2

After the binary geometry file has been created, it can be used as input, along with the hydrodynamic boundary conditions, to generate a hydrodynamic solution, or flow field, over the model domain. Either a steady state or dynamic flow field can be used with *SED2D-WES*. When using a dynamic flow field, an initial steady state flow field should be generated, using the boundary conditions that will be applied to the first time step of the dynamic simulation. This additional steady state simulation can be used to generate a hotstart of suspended sediment concentrations before running the *SED2D-WES* analysis.

A basic assumption of *SED2D-WES* is that the flow field will not significantly change with small amounts of deposition/erosion. At the end of each time step, *SED2D-WES* saves a new ASCII geometry file (_out.geo), which has the accumulation of deposition or erosion added into the bathymetry of each node. At times, large changes in the bed occur, which require the flow field to be recomputed with a new geometry. By default, *SED2D-WES* stops when the deposition or erosion

at any node exceeds 25% of the original water depth. If such a condition occurs, the *RMA2* simulation should be restarted where *SED2D-WES* stopped, using the updated geometry to compute a more accurate flow field. Using the new flow field, the *SED2D-WES* simulation can be continued.

9.1.3 SED2D-WES

Part of the *SED2D-WES* simulation definition includes a user-assumed suspended sediment concentration over the entire model domain. As the simulation starts, the initial concentration at each node is quickly adjusted, based on computations of the governing equations. Such adjustments cause an artificially high deposition/erosion rate until the suspended sediment concentrations settle down to model-expected values. To keep these adjustment effects out of the final simulation, *SED2D-WES* should be run using an initial steady state hydrodynamic solution and constant sediment boundary conditions equivalent to the desired starting condition. The solution generated can be used to hotstart the suspended sediment concentrations.

9.2 RMA2 Coldstart Simulation

The hydrograph to be used with the *SED2D-WES* simulation is accompanied by a rating curve to define the downstream water surface. Before running the hydrograph analysis, a steady state solution will generate initial velocity and suspended sediment concentration values.

- Open the simulation *s.sim*.

This simulation has the geometry and material properties set up. To find out more about setting up these entities, see the tutorials that discuss *RMA2* simulations. The geometry has a constant slope of 0.002. The upstream boundary is at the top left cross section while the downstream boundary is at the bottom right cross section.

As was previously stated, the downstream water surface will be determined using a rating curve. However, it is often difficult to run a coldstart simulation with a rating curve. Before running the steady state rating curve simulation, a coldstart will be run using a head boundary condition on the downstream end.

From the rating curve, a flowrate of 3000 cms produces a downstream water surface elevation of about 10.0 meters. Notice, however, that the highest nodal bathymetry value is above 18.0 meters. An *RMA2* coldstart simulation must start with the downstream water surface set higher than all the node bathymetry values or the simulation most likely will not converge. This downstream head for this coldstart simulation will be set to start at 30 meters and slowly lower to 10 meters.


To set up global parameters for this initial steady state simulation:

1. Select *RMA2 / Model Control* to open the *RMA2 Model Control* dialog.

2. Be sure to run a *Steady state* simulation.
3. Turn on the *Hotstart output file* option.
4. Set the *Iterations* for the *Initial Solution* to 100 then turn on the *Steady state depth convergence* with a convergence value of 0.0005.
5. Click the *Optional BC Controls* button to open the *Optional BC Controls* dialog. Set the *Initial WSE* value to 30.0 and click *OK*.
6. Click *OK* from the *RMA2 Model Control* dialog.

Setting the number of iterations to a very high value and turning on the convergence parameter assures that *RMA2* converges before continuing to the next step. If *RMA2* does not reach convergence, the solution is invalid. The *Initial WSE* (water surface elevation) value is only used on a coldstart simulation and should be set to the downstream head value. It is applied to all nodes in the mesh as a starting point so that a solution can be obtained.

To define the initial boundary conditions:

1. Create a nodestring  across the upstream (top left) boundary. Be sure to create it from right to left while looking downstream.
2. Select the nodestring. Arrows should appear, pointing into the mesh. If the arrows point upstream, it was not created in the correct direction and you should choose the “Nodestring | Reverse Direction” menu command.
3. Select *RMA2 / Assign BC*. Choose the *Flow BC* option and assign a *Flowrate* of 3000 (cms) with the *Perpendicular to boundary* option. Then click *OK*.
4. Create a nodestring across the downstream (bottom right) boundary. Once again, be sure to create it from right to left when looking downstream.
5. Select the nodestring. This time, be sure the arrows point out of the mesh.
6. Select *RMA2 / Assign BC*. Select the *Head BC* option and assign an *Elevation* value of 30 (m). Then click *OK*.

The initial simulation for a constant downstream head of 30 meters has now been defined. However, we want to progressively lower this boundary condition to 10 meters. This can be done in *RMA2* using revisions, but revisions cannot be created inside *SMS*. You will manually define revisions after saving the simulation. To save this initial *RMA2* simulation:

- Choose *File / Save Project*. Enter the name “s-cold” and click *Save*.


Various files are saved with the project, including the initial *RMA2* simulation. You now need to edit the boundary condition file, adding in the required revisions to

lower the water surface to 10 meters. A revision consists of the REV card, followed by any data that needs to be revised. Each revision should be located before the END card that marks the end of the time step. In this case, only the BHL card will be updated during the revisions. See the *RMA2* help file for more information on these and other data cards. To add the revisions:

1. Choose *File / View Data File*. Open the file “s-cold.bc” and choose a text editor. Then click *OK*.
2. Add the following pairs of REV and BHL cards just before the END card (there should only be one END card because this is a steady state simulation).
3. Save the file and exit the text editor.

REV		
BHL	2	27
REV		
BHL	2	24
REV		
BHL	2	22
REV		
BHL	2	20
REV		
BHL	2	18
REV		
BHL	2	16
REV		
BHL	2	14
REV		
BHL	2	12
REV		
BHL	2	10


The last BHL card above should be directly followed by the END card. If you wish to compare your file with that created by the *SMS* developers, you can look in the tutorials\out directory under the *SMS* installation. This directory contains a copy of all files that you will create in the tutorial lessons. With the revisions added to the boundary conditions, you are ready to run the initial solution. To do this:

1. Select *RMA2 / Run GFGEN*. You should be shown the location of the gfgv435.exe file. Note: if you see the message “gfgv435.exe - <not found>”, click the *File Browser*  button and locate it (*GFGEN* should be in the *models* directory under the *SMS* installation). Click the *OK* button to run *GFGEN*.
2. When *GFGEN* is finished, press the *Enter* key in its window (it should only take a few seconds).
3. Select *RMA2 / Run RMA2*. Once again, find the executable if necessary and click the *OK* button.
4. When *RMA2* is finished, press the *Enter* key in its window (it takes about 1.5 minutes on an Intel 450 MHz system to execute all revisions).

RMA2 creates a solution named “s-cold.sol”, but this will not be used for anything. In addition, *RMA2* creates the file “s-cold.hot”, which is a hotstart point with a downstream water surface elevation of 10 meters.

9.3 RMA2 Rating Curve Simulation

The hotstart file saved from the previous *RMA2* run needs to be used as input to the rating curve simulation. To do this:

1. Select *RMA2 / Model Control* to open the *RMA2 Model Control* dialog.
2. Turn on the *Input hotstart file* option.
3. Click the *File Browser*  button and open the file “s-cold.hot”.
4. Click the *OK* button from the *RMA2 Model Control* dialog.

You are now ready to introduce the rating curve into the simulation. The rating curve definition consists of four values, which describe an exponential equation to compute the flowrate, and a fifth value to define the flow direction (see the BRC card in the *RMA2* help file). The following values will be used for this simulation:

$$A1 = 1450$$

$$A2 = 700$$

$$E0 = 8.0$$

$$c = 1.0$$

$$\text{Direction} = 270^\circ = 4.712389 \text{ radians}$$

In tests from the *SMS* developers, the rating curve definition has not worked when the *c* value is not 1.0, and other stability problems can occur. This presents a limitation on how well the rating curve definition can describe the physical situation. For this “laboratory” test case, such a limitation does not matter, but you should be aware of it before attempting to use a rating curve in a real-world simulation.

A rating curve for *RMA2* cannot be created inside *SMS* so you once again need to edit the boundary condition file after saving the simulation. First, you should save a new simulation. To do this:

1. Select *File / Save As* to save a new simulation (don’t select *Save Project*).
2. Enter the name “s-init” and click *Save*.

The boundary conditions created by *SMS* are an upstream flowrate and downstream head. The head boundary needs to be converted to a rating curve. It turns out that *RMA2* will not converge when the rating curve is added directly. It needs to be revised a few times to obtain the end condition. To add the rating curve:

1. Select *File / View Data File* and *Open* the file “s-init.bc” into a text editor.
2. Find the BHL line and make note of the nodestring number – it is the first number after the card identifier, and should be 2 if this tutorial has been explicitly followed. Remove this line from the file.
3. Add the following BRC and REV cards between the BQL and END cards, just as you did in the previous section. Be sure to use the nodestring number that was on the original BHL card as noted in the previous step (it should be 2).
4. Save the file and exit the text editor.

BRC	2	1440	500	8	1.00	4.712389
REV						
BRC	2	1440	550	8	1.00	4.712389
REV						
BRC	2	1440	600	8	1.00	4.712389
REV						
BRC	2	1440	625	8	1.00	4.712389
REV						
BRC	2	1440	650	8	1.00	4.712389
REV						
BRC	2	1440	675	8	1.00	4.712389
REV						
BRC	2	1450	700	8	1.00	4.712389

Note that there is no REV card before the first BRC card because it defines the initial rating curve used, not a revised rating curve. The last BRC curve has the final rating curve values that will be used with the full hydrograph simulation, and it should be directly followed by the END card. Once again, you may want to compare your file with that saved by the *SMS* developers. Now that the .bc file has been updated, you are ready to run this initial simulation. To do this:

1. Select *RMA2 / Run GFGEN* and click the *OK* button to run *GFGEN*. When *GFGEN* finishes, press the *Enter* key in its window.
2. Select *RMA2 / Run RMA2* and click the *OK* button. When *RMA2* is finished, press the *Enter* key in its window (it takes about 3 minutes on an Intel 450 MHz system to execute all revisions).

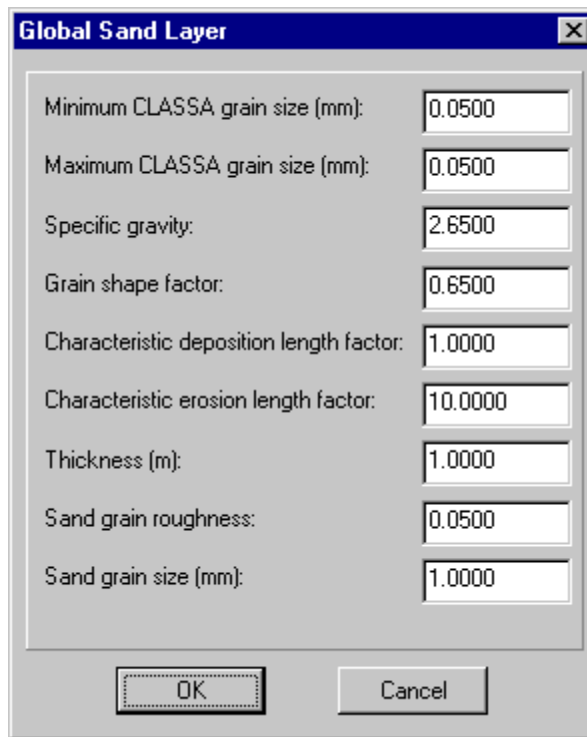
RMA2 creates a solution named “s-init.sol”, which will be used to define the flow field for the initial steady state *SED2D-WES* simulation. In addition, *RMA2* creates the file “s-init.hot”, which is a hotstart file for the full hydrograph simulation with a downstream rating curve.

9.4 Initial SED2D-WES Simulation

You now have a steady state *RMA2* solution whose boundary conditions are the same as the base flow condition for a hydrograph simulation. Before running the *RMA2* hydrograph analysis, you should run a *SED2D* simulation using this steady state flow field to generate initial suspended sediment concentrations throughout the mesh.

For this *SED2D-WES* simulation, a constant incoming suspended sediment concentration of 0.5 ppt (parts per thousand) will be used. The bed type is fine sand with an effective grain size of 0.05. To set up the bed information:

1. Select *SED2D / Global Parameters* to open the *SED2D Global Parameters* dialog.
2. Make sure the *Bed Type* is set to *Sand* and click the *Set Up Bed* button. Set the values as shown in Figure 9-2 and click the *OK* button. Note: for the current version of *SED2D-WES* (3.2), the *Minimum* and *Maximum* CLASSA grain sizes must be equal.
3. Set the *Diffusion coefficients* to 1.5. This determines how fast the sediment in suspension is distributed throughout the model.
4. Assign the *Initial concentration* as 0.5. This defines the initial suspended sediment concentration at all nodes in a *SED2D-WES* coldstart simulation. It is analogous to the *Initial WSE* value in *RMA2* to define the initial water surface elevation at all nodes of an *RMA2* coldstart simulation.
5. Change the *Settling velocity* to 0.001 (m/s).
6. Click the *OK* button to close the *SED2D Global Parameters* dialog.

A screenshot of a software dialog box titled "Global Sand Layer". It contains nine input fields with numerical values and two buttons at the bottom: "OK" and "Cancel". The values in the fields are: Minimum CLASSA grain size (mm): 0.0500, Maximum CLASSA grain size (mm): 0.0500, Specific gravity: 2.6500, Grain shape factor: 0.6500, Characteristic deposition length factor: 1.0000, Characteristic erosion length factor: 10.0000, Thickness (m): 1.0000, Sand grain roughness: 0.0500, and Sand grain size (mm): 1.0000.

Parameter	Value
Minimum CLASSA grain size (mm):	0.0500
Maximum CLASSA grain size (mm):	0.0500
Specific gravity:	2.6500
Grain shape factor:	0.6500
Characteristic deposition length factor:	1.0000
Characteristic erosion length factor:	10.0000
Thickness (m):	1.0000
Sand grain roughness:	0.0500
Sand grain size (mm):	1.0000

Figure 9-2. The Sand bed values to assign.

The next step to define the *SED2D-WES* simulation is to set up the equation and time controls. To do this:

1. Select *SED2D / Model Control* to open the *SED2D Model Control* dialog.
2. Set the *Hydraulic Bed Shear Stress* to use *Manning equation*.
3. Assign the *Time step length* as 0.5 hours, *Simulation time* as 24.0 hours, and the *Number of cycles* as 48 (maximum number of time steps to run).
4. Click the *OK* button to close the *SED2D Model Control* dialog.

The final part of setting up the *SED2D-WES* simulation is to assign the suspended sediment concentration at all inflow boundary conditions. This concentration is specified in ppt, parts per thousand. To assign the suspended sediment concentration:

1. Select the inflow nodestring and choose *SED2D / Assign BC*.
2. Assign a *Constant* concentration of 0.5 (ppt) and click *OK*.

The *SED2D-WES* simulation is now ready to be saved. To do this:

1. Select *File / Save As* (not *Save Project*).

2. From the *Save as type* list, choose the '*SED2D Control (*.sed)*' type. You should save only the *SED2D-WES* data (not the entire project) or you would write over the changes you just made to the .bc file.
3. Enter the name "s-init.sed" and click the *Save* button.

With the initial simulation saved, you are now ready to run *SED2D-WES*. This simulation uses the binary geometry and solution files that were created in the previous section. To run this simulation:

1. Select *SED2D / Run SED2D* and click the *OK* button to run *SED2D*. Once again, if the location of *sed2dv32.exe* is unknown, find it manually.
2. Click the *OK* button (it takes about 45 seconds on an Intel 450 MHz system). When *SED2D-WES* finishes, press the *Enter* key in its window.

The file "s-init_dbed.sol" is created and will be used to hotstart the suspended sediment concentrations for the hydrograph analysis. Note that this is the *SED2D-WES* solution file and it can be opened inside *SMS*, but this would not have significant meaning at this point of the analysis.

9.5 RMA2/SED2D Hydrograph Analysis

You have created an *RMA2* solution that has base flow conditions before the hydrograph is introduced. You also have a *SED2D-WES* solution that has suspended sediment concentrations for the *RMA2* solution. You are now ready to introduce the hydrograph and obtain the dynamic flow field solution so that you can run the true *SED2D-WES* simulation.

9.5.1 Updates For RMA2

The first thing to do is update the *RMA2* model controls. To do this:

1. Select *RMA2 / Model Control* to open the *RMA2 Model Control* dialog.
2. Change the *Solution type* to *Dynamic*.
3. Set the *Time step* to 0.5 (hours) and the *Num time steps* to 48 so that the *RMA2* simulation models one full day.
4. Increase the *Iterations* for *Each time step* to 100 and turn on the *Dynamic depth convergence* with a value of 0.0005 to assure convergence of *RMA2* at each time step.
5. Change the *Hotstart input file* to "s-init.hot" instead of "s-cold.hot".


6. Click the *OK* button to accept these changes.

Now that a dynamic simulation is set up, you need to assign a hydrograph at the upstream boundary. A time series file was previously saved which defines the hydrograph values for various hour intervals. To assign this boundary condition:

1. Select the upstream flow nodestring and choose *RMA2 / Assign BC*.
2. Change the boundary condition to *Transient*.
3. Click the *Define Curve* button. Click the *Import* button and *Open* the file “hydrograph.xls”. Then click *OK*. The assigned hydrograph is shown.
4. Click the *OK* button to assign the new boundary condition.

9.5.2 Updates For SED2D

The previous *SED2D-WES* data should still be set up. However, the previous simulation did not use an input hotstart file for suspended sediment concentrations. You need to tell *SED2D-WES* that this simulation does use an input hotstart file. To do this:

1. Select *SED2D / Global Parameters* to open the *SED2D Global Parameters* dialog.
2. Under the *Hotstart Options* section, turn on *Concentrations*, then click the *File Browser*  button and open the file “s-init_dbed.sol”.
3. Click the *OK* button.

9.5.3 Saving the Project

To save this new project:

1. Select *File / Save As* (do not select *Save Project*).
2. Enter the name “s-hydro” and click the *Save* button.

Before running the simulation, replace the *BHL* card in the boundary condition file (s-hydro.bc) with the *BRC* card shown below (once again, be sure to retain the correct nodestring id).

```
BRC  2  1450  700  8  1.00  4.712389
```

9.5.4 Running the Simulation

After saving the project and making the required changes to the saved data files, you are ready to run the simulation. To do this:

1. Select *RMA2 / Run GFGEN* and press the *OK* button to run *GFGEN*. When *GFGEN* finishes, press the *Enter* key in its window.
2. Select *RMA2 / Run RMA2* and press the *OK* button to run *RMA2* (it takes about 9.5 minutes on an Intel 450 MHz system to execute all time steps). When *RMA2* finishes, press the *Enter* key in its window.
3. Select *RMA2 / Run SED2D* and press the *OK* button to run *SED2D-WES*. When *SED2D-WES* finishes, press the *Enter* key in its window.

After running the simulation, there are two solution files that can be opened into *SMS*: the hydrodynamic *RMA2* solution “s-hydro.sol” and the sediment transport *SED2D-WES* solution “s-hydro_dbed.sol”. In addition, *SED2D-WES* saves a new ASCII geometry file named “s-hydro_out.geo”, which contains updated bathymetry values at all nodes in the mesh. This is in the same format as the geometry file originally saved by *SMS*.

For tips and assistance on viewing and interpreting the simulation results, see the tutorial lessons that discuss post-processing techniques.

9.6 Conclusion

This concludes the *SED2D-WES* Analysis tutorial. You may continue to experiment with *SMS* or you may close the program. To exit *SMS* at this time:

- Select *File / Exit*. Click the *Yes* button if asked to confirm.

RMA4 Analysis

10.1 Introduction

This lesson will teach you how to run a solution using *RMA4*. If you have not yet completed Lesson 4 on *RMA2*, you should do so now. *RMA4* is part of the TABS-MD suite of programs and is used for tracking constituent flow in 2D models. In this lesson, you will use *RMA4* to model 3 situations: an inflow of a constituent into a river, the inflow of a constituent into a bay, and salinity intrusion.

10.2 Case 1

RMA4 can only be run after having initially run a solution in *RMA2*. This is because *RMA4* uses the flow solutions computed by *RMA2* to compute the constituent concentration as it flows through the mesh. An *RMA2* geometry and solution have been supplied. To open the *RMA2* files:

1. *Select File / Open.*
2. Select the file *madora.spr* from the *tutorial\tut10* directory. If you still have geometry open from a previous tutorial, you will be warned that all existing data will be deleted. If this happens, click the *OK* button.

The geometry will be displayed on the screen with the *RMA2* boundary conditions.

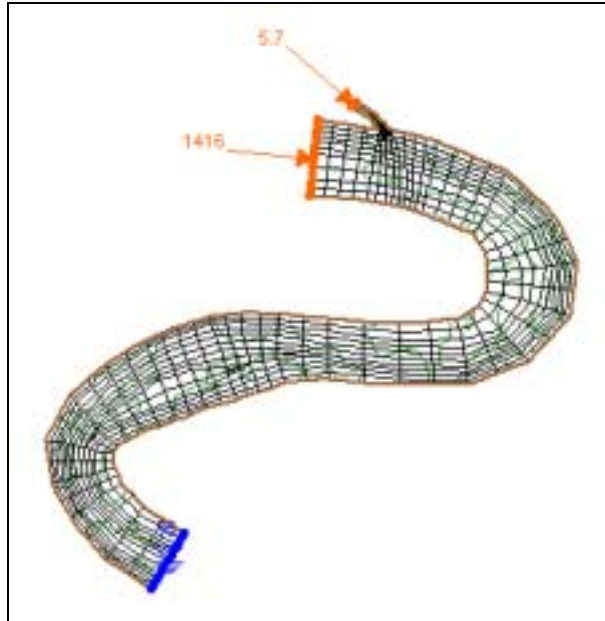


Figure 10-1 Madora mesh

The mesh was created in Metric units because *RMA4* requires Metric units. The main channel has a flow of 1,416 cms (50,000 cfs) with a channel entering with a flow of 5.7 cms (2,000 cfs).

10.2.1 RMA4 Model Control


RMA4 is a transient model. The *RMA2* solution is a steady state solution. *RMA4* will assume a steady flow throughout the mesh, but the boundary conditions will change. To set the time that *RMA4* will run:

1. Go to *RMA4 / Model Control*.
2. Set the times as follows:
 - *Start time*: 0.0
 - *Time step*: 0.5 (hrs)
 - *Total time steps*: 50
 - *Max time*: 24 (hrs)
 - *Last time step from the RMA2 velocity file to be used*: 0.0 (hrs)
 - *Number of hours subtracted from the RMA2 velocity file*: 0.0 (hrs)

3. Make sure the *Geometry input file*, *Flow input file*, *Binary output file*, and *Full output options* are on and the other *File Control* options are off.
4. Push the *Advanced Output Control* button, turn the *Print all* command on, and push *OK*.
5. Push *OK* to exit the *RMA4 Control* dialog.

10.2.2 Boundary Conditions

For this model, a pollutant has been dumped into the smaller channel for three hours. The concentration of the pollutant in the stream is 1,000 ppm. To apply this boundary condition:

1. Select the *Select Nodestring*  tool from the *Toolbox*.
2. Select the nodestring at the smaller inflow boundary (labeled as 5.7).
3. If the arrows are not pointing into the larger channel select the *Nodestrings / Reverse Direction*.
4. Select *RMA4 / Assign BC*.
5. Switch to *Transient* and push the *Define Curve* button.
6. The *XY Series Editor* dialog appears. In this dialog, a time series curve can be created. To turn the pollutant on for only 3 hours:
 - a. Click the *New* button.
 - b. Enter the following *Time/Concentration* values:

Time	Concentration
0.0	1000.0
3.0	1000.0
3.1	0.0
24.0	0.0
 - c. Push *OK* to exit the *XY Series Editor*.
7. Push *OK* to exit the *RMA4 Assign BC* dialog.

NOTE: In this case, we applied 1,000 ppm as a boundary condition. *RMA4* does not care about the units of the concentration. The output from *RMA4* is relative to the initial number you specify. For example, we specified a concentration of 1,000. The

values in the solution will range from 0 to 1,000 as the plume spreads downstream. We can say that the concentration was ppm, ppt, or kg/kg; RMA4 treats all concentrations as relative values.

10.2.3 Material Properties

The final step is to specify the Material diffusion. To do this:

1. Select *RMA4 / Material Properties*.
2. Select each material, turn on the *Material diffusion*, and set the *X* and *Y diffusion coefficients* to 10.0 (m²/s).
3. Push *Close* to exit the dialog.

Because *RMA4* does not have the ability to model turbulence, diffusion coefficients may be used to approximate turbulence. By assigning a diffusion coefficient in the *x* and *y* directions for each material, the flow over that material will be altered somewhat to provide an approximation of turbulent flow over that region. A value of -1.0 may be applied (this is the default) to allow normal flow over the material. Positive values provide turbulence. The higher the value, the greater the effect is.


10.2.4 Run RMA4

You are now ready to run *RMA4*. Before running the model, you will need to save the data:

1. Select *File / Save as*.
2. Set the file type to *Project Files* and enter *madora* as the filename.
3. Click *Save* to save the file.

NOTE: *RMA4* requires that the *RMA2* and *RMA4* filenames be the same, so you must save the project as *madora.spr*.

To run *RMA4*:

1. Select *RMA4 / Run RMA4*.
2. If the prompt shows a message that *RMA4* is *not found*, click the *File Browser*  button manually find the correct program executable.
3. Click the *OK* button to launch *RMA4*.


RMA4 will take a few minutes to run and will beep when it is done. The file *madora.qsl* is the solution file containing the constituent data at each node.

10.2.5 Film Loop

Once a solution has been created by *SMS*, you can use a number of features to view the results and adjust the model to better approximate the observed values. The easiest way to view the results from the *RMA4* solution is to use the film loop. Before using the film loop, the *RMA4* solution file must be imported:

1. Select *Data / Data Browser*.
2. Click the *Import* button, select *madora.qsl*, and push *Open*.
3. Push *Done* to exit the *Data Browser*.

To create the film loop:

1. Select *Data / Film Loop* and push the Setup button.
2. Under *Data Options* in the bottom right section, click the *Scalar data set* button.
3. Click the *Display Options* button .
4. In the *Display Options*, turn off everything but *Elements* and *Contours*.
5. Click the *Options* button to the right of *Contours*.
6. Click the *Color fill between contours* button, turn on the *Contour between specified range*, set the *Minimum value* to 0.0 and the *Maximum value* to 10.0. (This is done because when the stream flow enters the main channel, the concentration quickly drops to between 0.0 and 10.0 ppm.)
7. Push *OK* twice to get back to the *Film Loop* dialog.
8. Push *OK* to start generating the film loop. Each frame of the film loop will be generated. After the film loop is done generating, click the *Play* button to view the film loop.
9. Click *Done* when you are finished watching the results.

You will notice how the concentration drops quickly as the pollutant enters the main channel. This occurs because the inflow from the small channel is 0.4% of the inflow from the main channel.

10.3 Case 2

In this case, a constituent is coming into Noyo Bay from a river. The files for this case can be opened by:

1. Select *File / Open*.
2. Select the file *noyo1.spr* from the *tutorial\tut10* directory. If you still have geometry open from a previous tutorial, you will be warned that all existing data will be deleted. If this happens, click the *OK* button.

The geometry will be displayed on the screen with the *RMA2* boundary conditions.

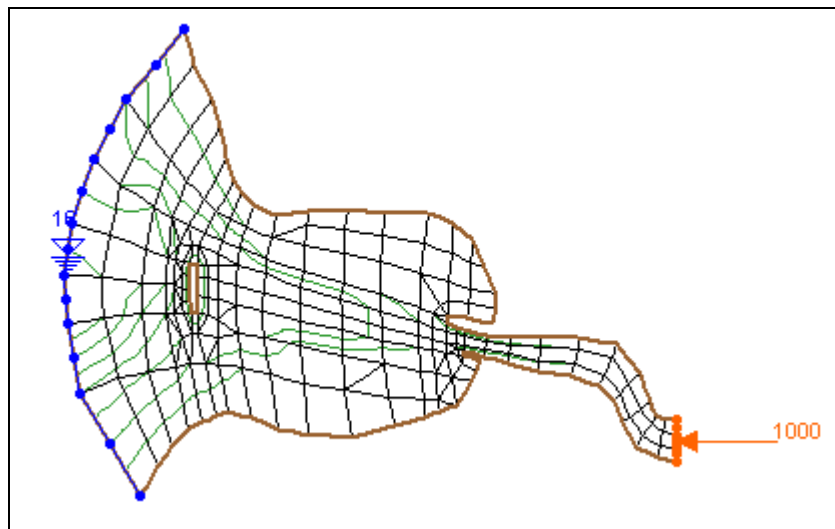


Figure 10-2 Noyo mesh

This mesh was created in English units. The river flowing into Noyo Bay has a flow of 1,000 cfs. The water surface elevation on the left side varies as the tide comes in and out over a 12-hour cycle repeated twice a day.

10.3.1 RMA4 Model Control

To set the model times:

1. Go to *RMA4 / Model Control*.
2. Set the times as follows:
 - *Start time*: 0.0
 - *Time step*: 0.5 (hrs)
 - *Total time steps*: 49


- *Max time*: 24 (hrs)
- *Last time step from the RMA2 velocity file to be used*: 24.0 (hrs)
- *Number of hours subtracted from the RMA2 velocity file*: 12.0 (hrs)

The last two times will cause *RMA4* to use the last 12 hours of the *RMA2* solution.

3. Make sure the *Geometry input file*, *Flow input file*, *Binary output file*, and *Full output options* are on and the other *File Control* options are off.
4. Push the *Advanced Output Control* button, turn the *Print all* command on, and push *OK*.
5. Push *OK* to exit the *RMA4 Control* dialog.

10.3.2 Boundary Conditions

For this model, a constant inflow of 100 ppm of a pollutant enters the bay from the river. To apply this boundary condition:

1. Select the *Select Nodestring*  tool from the *Toolbox*.
2. Select the nodestring at the right side of the model and select *RMA4 / Assign BC*.
3. Set a *Constant Concentration* of 100.0 (ppm) and push *OK* to exit the *RMA4 Assign BC* dialog.

10.3.3 Material Properties

To apply the diffusion:

1. Select *RMA4 / Material Properties*.
2. Select each material, turn on the *Material diffusion*, and set the *X* and *Y diffusion coefficients* to 1.0 (m²/s).
3. Push *Close* to exit the dialog.

10.3.4 Run RMA4

Save and run *RMA4* as you did for Case 1. Save the project as *noyo1.spr*. As mentioned earlier, the Noyo mesh was run using English units and *RMA4* requires

metric units. When you save the file, *SMS* checks to see what the current units are set as, which are English in this case. If the units are English, *SMS* writes the HS and GS cards to the .trn file to tell *RMA4* to convert to metric units.

Run *RMA4* as done earlier. After *RMA4* runs, the solution file will be named *noyo1.qsl*. Import the file using the Data Browser.

10.3.5 Film Loop

Generate a film loop using the same steps as for case 1 (Section 10.2).

10.4 Case 3

For this final case, we will view salinity intrusion into Noyo Bay. Open *noyo2.spr*.

10.4.1 RMA4 Model Control

To set the model times:


1. Go to *RMA4 / Model Control*.
2. Set the times as follows:
 - *Start time*: 0.0
 - *Time step*: 0.5 (hrs)
 - *Total time steps*: 49
 - *Max time*: 24 (hrs)
 - *Last time step from the RMA2 velocity file to be used*: 24.0 (hrs)
 - *Number of hours subtracted from the RMA2 velocity file*: 12.0 (hrs)

The last two times will cause *RMA4* to use the last 12 hours of the *RMA2* solution.

3. Make sure the *Geometry input file*, *Flow input file*, *Binary output file*, and *Full output options* are on and the other *File Control* options are off.
4. Push the *Advanced Output Control* button, turn the *Print all* command on, and push *OK*.
5. Push *OK* to exit the *RMA4 Control* dialog.

10.4.2 Boundary Conditions

For this model, a constant inflow of 100 ppm of a pollutant enters the bay from the river. To apply this boundary condition:

1. Select the *Select Nodestring*  tool from the *Toolbox*.
2. Select the nodestring at the **left** side of the model and select *RMA4 / Assign BC*.
3. Set a *Constant Concentration* of 8.0.
4. Select the *Factor applied when flow direction changes* button and set the *Shock Factor* to 0.5.
5. Push *OK* to exit the *RMA4 Assign BC* dialog.

Since a concentration in water is rarely rigidly maintained, a shock factor may be applied to allow fluctuation of the concentration when the flow direction changes. If no shock factor is applied, no matter how much the flow pushes the concentration out of the model, the concentration at the boundary will not change. However, applying a shock factor is like creating a buffer zone outside the model where the constituent can go until the flow begins to carry it back into the model. This provides for a more realistic solution in some cases. Depending on the situation, a different shock factor may be applied from zero for no shock to 1.0 for a gradual change due to a change in flow direction.

10.4.3 Material Properties

To apply the diffusion:

1. Select *RMA4 / Material Properties*.
2. Select each material, turn on the *Material diffusion*, and set the *X* and *Y diffusion coefficients* to 1.0 (m²/s).
3. Push *Close* to exit the dialog.

10.4.4 Run RMA4

Save and run *RMA4* as you did for Cases 1 and 2. Save the project as *noyo2.spr*.

Run *RMA4* as done earlier. After *RMA4* runs, the solution file will be named *noyo2.qsl*. Import the file using the Data Browser.

10.4.5 Film Loop

Generate a film loop using the same steps as for case 1. The only difference will be in the *Film Loop Options* dialog (after pushing *Setup*):

- In the bottom right of the dialog, set the *Run simulation from time* to 6.0 and the *Run simulation to* 18.0 hours.

Running these times will show a full tidal cycle that runs continuously.

10.5 Other Changes

You may want to play with the shock factor and diffusion coefficients to see how they affect the model. Other options include:

- Change the diffusion coefficients in all 3 cases to 0.5 and then try 10.0 to see the differences.
- Change the shock factor in the third case to 0.0 and 1.0. There is a large difference in how far the intrusion gets into the bay.

10.6 Conclusion

This concludes the *RMA4* Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File / Exit*. If prompted to confirm, click the *Yes* button.

HIVEL Analysis

11.1 Introduction

This lesson will teach you how to prepare a mesh for and run a solution using *HIVEL2D*. You will be using the file *proto.sup*, which references a finite element mesh stored in *proto.geo*. To open the file:

1. Select *File | Open*.
2. Find and highlight the file *proto.sup*. If you still have geometry data open from a previous tutorial, you will be warned that all existing mesh data will be deleted. If this happens, click *OK* to the prompt. The geometry data will open as shown in Figure 11-1.

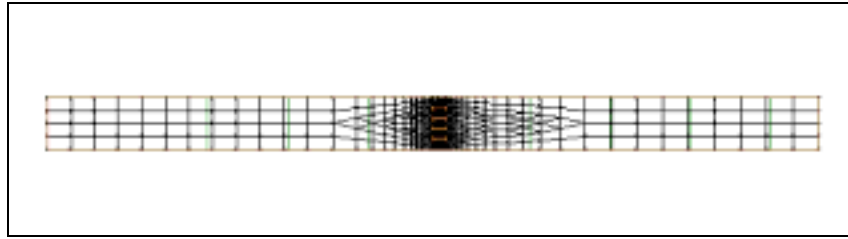


Figure 11-1 The mesh contained in the file *proto.geo*.

11.2 Creating Materials

When this mesh was opened, the elements each contained a material type ID. The materials were created with default parameters that must be changed for this particular mesh. The material properties define the roughness for the elements.

There is only one material in this example. To edit the material parameter:


1. Select *HIVEL | Material Properties*.
2. In the *HIVEL Material Properties*, select the material labeled *material 01*. Be sure that its ID number is 1 in the top right corner.
3. Enter 0.014 for *n* (Manning's roughness).
4. Click the *Close* button.

The material now has the correct parameter associated with it. (Materials can be displayed by opening the *Display Options* dialog and turning on the *Materials* toggle under the 2D Mesh tab.)

11.3 Creating Nodestrings

Before boundary conditions can be applied at the inflow or outflow boundaries, nodestrings must first be created.

To create the nodestrings:

1. Choose the *Create Nodestrings*  tool from the *Toolbox*.
2. To create the inflow nodestring, click on the node on the top left side of the mesh. While holding *Shift* key, double-click on the node on the bottom of the left side.
3. Repeat step 2 to create the outflow nodestring on the right side of the mesh.

11.4 Defining Boundary Conditions

11.4.1 General Parameters

The finite element network is only the first part of the numerical model. We have already defined the material properties associated with the elements in the mesh. In addition to the geometry and the material properties, we must define several other model parameters. To edit the *HIVEL2D* parameters:


1. Select *Edit / Current Coordinates* and make sure the *Horizontal System* is set to *Local* and both *Units* are set to *U.S. Survey Feet*. Click *OK*.
2. Select *HIVEL / Model Control*.
3. Enter the following values (select English units in step 1 first because changing units will automatically update system variables such as gravity):
 - Max # of iter./timestep = 6
 - Time step size = 4.0
 - # time steps = 100
 - Save output every = 100th timestep
4. Push *OK* to exit the dialog.

HIVEL2D simulates steady state boundary conditions but uses pseudo-dynamic analysis to allow the flow to change from the initial conditions to the final boundary conditions.

11.4.2 Defining Steady State Flow and Head


For this tutorial, we will define boundary conditions at the nodestring of the inflow boundary, as well as along a nodestring at the outflow boundary.

To assign the inflow boundary condition:

1. Choose the *Select Nodestrings*  tool from the *Toolbox* and select the nodestring across the inflow boundary on the left side of the mesh.
2. Select *HIVEL / Assign BC*.
3. Select the *Supercritical* and *Inflow string* options.

4. Select *unit discharge*, *x and y components*, and *depth* for the *Inflow Parameters*.
5. Enter 148.5 for *P* and 0.0 for *Q* (x and y components of unit flow), and 7.22 (ft) for *depth*.
6. Click the *OK* button or press the *Enter* key to leave the *Boundary Conditions* dialog.

To assign the outflow boundary condition on the right side of the mesh:

1. Choose the *Select Nodestring*  tool from the *Toolbox* and select the nodestring across the outflow boundary.
2. Select *HIVEL / Assign BC*.
3. Select the *Supercritical* and *Outflow string* options.
4. Click the *OK* button or press the *Enter* key to leave the *Boundary Conditions* dialog.

For *HIVEL2D*, flow is specified at the inflow or upstream boundaries and water surface elevation (head) is specified at the outflow or downstream boundaries for subcritical flow and at upstream boundaries for supercritical flow. If a jump occurs, you need to specify head at both boundaries. See the *HIVEL2D Reference Manual* for more about assigning boundary conditions.

11.4.3 Creating the Hotstart File

HIVEL2D must have an initial hotstart file to run a solution. *SMS* allows you to create this hotstart file either using constant values or a data set that has been previously loaded through the *Data Browser*. For this tutorial, a data set will be created using the *Data Calculator*.

1. Select *Data | Data Calculator*.
2. Enter “Start Depth” in the *Result* field.
3. Enter 7.22 in the *Expression* field. This is the constant depth.
4. Click the *Compute* button to add “Start Depth” to the Data Sets.
5. Click the *Done* button to leave the *Data Calculator* dialog.

Now the *HIVEL2D* hotstart file can be created.

1. Select *HIVEL | Build Hot Start*.

2. Enter 0.0 for the *Time associated with step m*.
3. For both *Step m-1* and *Step m*, select the *Constant* option for the discharge. Enter a value of 148.5 for *p* and 0.0 for *q* for both steps.
4. For *Step m-1*, select the *Data Set* option for the *Water Depth*. Press the *Select* button.
5. From the *Select Data Set* dialog highlight the “Start Depth” and press the *Select* button to exit.
6. Repeat steps 4 and 5 for *Step m*.
7. Push the *Write Hotstart Now* button.
8. Click the *OK* button to close the *Build Hot Start* dialog.

When *HIVEL2D* starts, it will use these values as an initial starting place. It is important to note that at the end of the computations, this file will be over written. Therefore, a backup copy may be created if desired. This is done by making a copy of the file using Windows Explorer. The file name is *hivel.hot* and was saved in the same directory where the geometry file was opened, in *tutorial\tut11*.

11.5 Saving The Simulation

Before saving the analysis, check the model for completeness. To do this:

1. Select *HIVEL | Model Check*.
2. Click the *Run Check* button.


The *HIVEL2D* model checker may report a possible warning. The warning is that the mesh has not been renumbered during this session. This can be ignored because it was done in a previous session.

HIVEL2D uses a geometry file, boundary condition file, and hotstart file written by *SMS* to run an analysis. These files are specified in a *super file* that is also written by *SMS*. You must save data that has been created.

1. Select *File | Save As* and make sure the *Save as type* is set to *Project Files*.
2. Enter *proto_h* for the *File name* and press *Save*.

11.6 Using HIVEL2D

The analysis program *HIVEL2D* can be launched from inside *SMS*. To do this:

1. Select *HIVEL / Run HIVEL*.
2. If *SMS* shows a message that the *hivel2d* executable is not found, click the *File Browser*  button to find the *HIVEL* executable.
3. Click the *OK* button to run the model.

HIVEL2D may take a few minutes to run this solution. When it is finished, the program may ask you to press *Return* before the window will go away. The model has simulated 400 seconds of flow time (100 time steps @ 4 seconds each). The run has created 3 new files:

- *proto_hflo.dat*: contains velocity data at each node.
- *proto_hwse.dat*: contains water surface elevation data at each node.
- *hivel.hot* – contains hotstart data to continue where *HIVEL2D* left off.

These files can be opened through the Data Browser.

11.7 Conclusion

This concludes the *HIVEL* Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File / Exit*. If prompted to confirm, click the *Yes* button.

CGWAVE Analysis

12.1 Introduction

This lesson will teach you how to prepare a mesh for analysis and run a solution for *CGWAVE*. You will start with the data file *indiana.xyz* which contains a set of points that contain depth data from which a mesh will be created. To open the data:


1. Select *File / Open*.
2. Select *indiana.xyz* in the *tutorial\tut12* directory and click the *Open* button.
3. The *File Import Wizard* dialog will appear. The settings are defaulted for opening an XYZ file. Push *Finish* to open the file. (This wizard allows you to open data that may not have data in 3 columns of x, y, and z. Data in any number of columns in any order can be opened through the wizard).

A gridded scatter set will be created.

12.2 Creating a Wavelength Function

The first step in creating a mesh for *CGWAVE* is to create a wavelength function. The wavelength function is an intermediate step to creating a size function, which is explained in section 12.3. The z value of each point in the *indiana.xyz* data is actually a water depth value. The wavelength at each point is calculated from this depth value using a complicated equation. It is sufficient to say that a larger

wavelength is calculated from a larger water depth value. To create the wavelength function:

1. Make sure you are in the *Scatter*  module.
2. Select *Data / Create Data Sets*.
3. Turn off everything but the *Transition Wavelength/Celerity* option. (The *Coastal* option must be on to access the *Transition Wavelength/Celerity* option.)
4. Leave the function name as *Transition* and enter a *Period* of 20.
5. Click the *OK* button.

Two new data sets will be created, one named *Transition_Wavelength* and the other named *Transition_Celerity*. These can be seen in the *Data Browser* (see the *SMS Help* for more information on the *Data Browser*).

12.3 Creating a Size Function

The size function is created from the wavelength function. The size function is the function that determines the element size that will be created by *SMS*. Each point is assigned a size value. This size value is the approximate size of the elements to be created in the region where the point is located. The mesh will be denser where the size values are smaller.

The wavelength function that was created in section 12.2 contains values that are twice as large as the desired size values. The wavelength function will be scaled by one half to create the size function. To do this:

1. Select *Data / Data Calculator*.
2. In the top left section of the *Data Calculator*, highlight the function named *Transition_Wavelength* and click the *Add to Expression* button. The letter that represents this function will appear in the *Expression* field.
3. Click the */* (divide symbol) in the middle section of the *Data Calculator*.
4. After the divide symbol, enter the number 2 (two) using the keyboard.
5. In the *Result* field, enter the name *size* and then click the *Compute* button.
6. When the computation is completed, *size* will appear as a data set.
7. Click the *Done* button to exit the *Data Calculator*.


A new data set named *size* is created which is half the *Transition_Wavelength* data set.

12.4 Defining The Domain


A domain represents the region that is offshore. In *CGWAVE*, the domain can be a circular, semi-circular, or rectangular region. In *SMS*, a *Feature Arc* is used to define the coastline. After the coastline is defined, *Feature Arcs* and *Feature Polygons* are used to define the domain region.

12.4.1 Creating the Coastline

SMS can automatically create a coastline at a specific elevation or water depth from a scattered data set. The active function of the active scattered data set will be used for this operation. You should currently have only one scattered data set. To make the elevation function active:

1. Make sure you are in the *Scatterpoint*  module.
2. Change the function in the *Scalar* field in the top *Edit Window* (just beneath the menus) from *size* to *elevation*.

Before creating the coastline, you should create a *CGWAVE* coverage. To do this:

1. Switch to the *Map*  module.
2. Select *Feature Objects / Coverages*.
3. Change the Coverage type to *CGWAVE* and click the *OK* button.


With the coverage type set and the active scatterpoint data set defined, you are ready to create the coastline. To create the coastline arc:

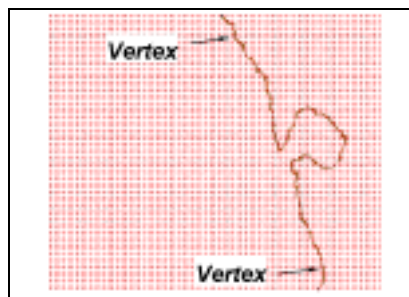
1. Select *Feature Objects / Create Coastline*.
2. You will be prompted for an elevation value. Enter the value of 1.0 and click the *OK* button to this prompt.

After a few seconds, the display will refresh with an arc representing the 1.0 water depth line, as shown in Figure 12-1a.

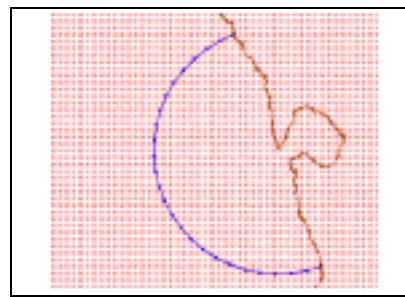
12.4.2 Creating the domain

With the coastline created, you are ready to create the mesh domain. This model will use a semi-circular domain that intersects with the coastline. To create the domain:

1. Choose the *Select Feature Vertex*  tool from the *Toolbox*.
2. Hold the *SHIFT* key and click on two vertices, one at each end of the coastline arc, as shown in Figure 12-1a.
3. Select *Feature Objects / Define Domain*. Select *Semi-circular*, and click *OK*. This creates a semicircular *Ocean* arc as shown in Figure 12-1b.



(a). The coastline feature arc.



(b). The domain feature arc.

Figure 12-1 The indiana scatterpoint and feature data.

Now that feature arcs define the domain, a feature polygon must be created from the feature arcs. To create the polygon:

- Select *Feature Objects / Build Polygons*.


After this command is executed, polygons are formed from any set of arcs that form a closed loop. The screen will not refresh when polygons are built, so it may appear that nothing happened even though polygons were created. For this example, there should now be a single polygon made from the semi-circular ocean arc and the part of the coastline arc with which it intersects.

12.5 Creating The Finite Element Mesh

There are various automatic mesh generation techniques that can be used to create elements inside a specified boundary. One of these is applied to each polygon, after which a finite element mesh can be generated. For this tutorial, there is only one polygon, which will be assigned the *Density mesh* type.

12.5.1 Setting Up The Polygon

When using density meshing, *SMS* determines element sizes from a *size function* in a scattered data set. The size function to be used in this example was created back in section 12.3. To set up the feature polygon for density meshing:

1. Choose the *Select Feature Polygons*  tool from the *Toolbox*. With this tool selected, double-click inside the polygon that defines the domain.
2. In the *Polygon Attributes* dialog, change the *Mesh Type* to *Scalar Paving Density* and press the Options button.
3. Change the *Density Type* to *Scalar Paving* (see the *SMS Help* for a description of the different meshing types).
4. In the bottom left of the *Scatter Options* dialog under *Mesh Type*, turn on the *Truncate values* option and set the *Min* and *Max* to 10 and 1000. This sets up a minimum and maximum size to be used when creating elements.
5. Select size in the *Data Set* window.
6. Click the *OK* button to get back to the *Polygon Attributes* dialog.
7. In the *Bathymetry Type* section, select the *Scatter Set*.
8. Select the *Scatter Options* dialog under *Bathymetry Type*, highlight the function named *elevation* in the *Data Set* window as the scatter function and make sure the *Truncate Values* option is turned off. As mesh nodes are created, their elevation value will be assigned from the original water depth values that were read from the xyz file.
9. Click the *OK* button to close both dialogs.

The polygon is now set up to generate finite elements inside the boundary. When more than one polygon exists, the meshing attributes need to be set up for each of the polygons.

12.5.2 Generating The Elements

Since there is only one polygon in this example, you are ready to have *SMS* generate the finite element mesh from the defined domain. To create the mesh:

- Select *Feature Objects / Map->2D Mesh*.

After a few moments, the mesh will be created to look like the finite element mesh in Figure 12-2.

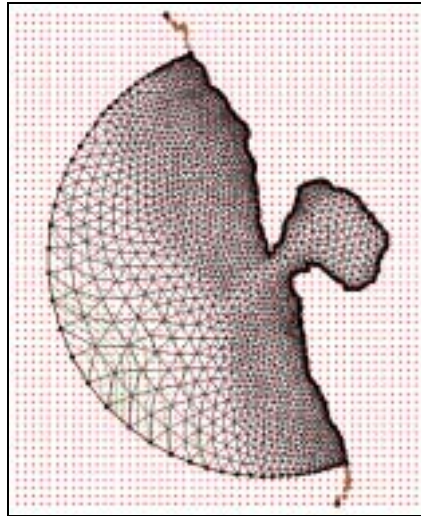





Figure 12-2 The completed finite element mesh.


At this point, the display is quite cluttered with all the data that has been created. By changing some display settings, the display can become less cluttered. To change the display settings:

1. Switch to the *Scatterpoint*  module and select *Display / Display Options*. Under the *Scatter* tab, at the top left there is a box labeled *Visible*, turn the *Visible* toggle off and click the *OK* button.
2. Switch to the *Map*  module and select *Feature Objects / Coverages*. Turn the *Visible* toggle off and click the *OK* button.
3. Switch to the *Mesh*  module and select *Display / Display Options*. Under the *2D Mesh* tab turn off everything except the *Elements* and *Contours*.
4. Click the *Contours* tab. Change the *Number of Intervals* to 20. Change the *Contour Method* to *Color fill*.
5. Click the *OK* button.

The display will refresh after steps 1, 2, and 5 above. After the final refresh of the display, you will see contours of water depth with the elements drawn on top of those. You can clearly see that as the water depth decreases, so does the element size. A dredged channel can be seen running into the harbor.


12.6 Model Control

When creating a *CGWAVE* model, the boundary conditions are wave amplitude, phase, and direction. To define these *incident wave conditions*:

1. In the *Mesh*  module, select *CGWAVE / Model Control*.
2. Set the *Incident Wave Conditions*: *Direction* = 30.0, *Period* = 20.0, and *Amplitude* = 1.0.
3. Set the *Number of Iterations* to 1 and the *Maximum Iterations* to 500,000.
4. CGWAVE uses a 1-d file. The 1-d parameters can be set in this dialog. By default, the # of 1-d nodes is set to 100. The 1-d Spacing is set to $1.25 \times \text{radius} / 100$. We can leave these defaults.
5. Click the *OK* button to exit the *CGWAVE Model Control* dialog.

12.7 Renumbering

The mesh needs renumbering before being saved. To do this:

1. Select the *Select Nodestring*  tool from the *Toolbox*.
2. Select the blue ocean nodestring by clicking in the box on the nodestring.
3. Select *Nodestrings / Renumber* and push *OK*.

12.8 Saving the CGWAVE Data


CGWAVE uses a geometry file and the 1-d file mentioned above to run an analysis. This file consists of two lines that run perpendicular from the coastline to the extents of the domain. The 1-d file is generated automatically by SMS using the active scatter set. The file contains depth information on both sides of the domain. To save these files:

1. Select *File / Save as*, make sure the Save as type is set to Project Files, and enter the name *indianaout.spr*.
2. Push the *Save* button.

12.9 Running CGWAVE

CGWAVE can be run from SMS. To run CGWAVE:

1. Select *CGWAVE / Run CGWAVE*.

2. Click on the  select file icon and find the *CGWAVE* executable. Click the *OK* button.

Respond to the prompts that appear as follows:

```
Input data filename: <*.dat>
indianaout.cgi
Output wave potential filename: <*.out>
indianaout.cgo
Input data filename for 1-D solution: <*.1d>
indianaout.cgl
A previous solution file as an initial state (y/n)?
n
```

For this simulation, *CGWAVE* should finish in a couple of minutes. When the simulation is finished, the file *indianaout.cgo* will contain the *CGWAVE* solution data.

NOTE: If *CGWAVE* does not run, you may have an older version of *CGWAVE*. Open the *indianaout.cgi* file in a text editor and change the first few lines from:

```
%number of characters in title &
%number of terms in the series &
%number of iterations for checking convergence &
%maximum iterations &
%maximum iterations for nonlinear mechanisms &
%maximum connectivity &
      12          35          1      500000          1000          8
```

to:

```
%number of characters in title &
%number of terms in the series &
%number of iterations for checking convergence &
%maximum iterations &
%maximum connectivity &
      12          35          1      500000          8
```

12.10 Conclusion

This concludes the *CGWAVE* Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File / Exit*. If prompted to confirm, click the *Yes* button.

ADCIRC Analysis

13.1 Introduction

This lesson will teach you how to prepare a mesh for analysis and run a solution for *ADCIRC*. It will cover preparation of the necessary input files for the *ADCIRC* circulation model and visualization of the output. You will start by reading in a coastline file and then a SHOALS file.

The data used for this tutorial is from Shinnecock Bay off of Long Island in New York. All files used for this tutorial are located in the *tutorial\tut13* directory.

13.2 Reading in a Coastline File

For this tutorial, you will first read in a coastline file, which has already been set up for you. This sample coastline will form the boundary for your mesh.



To open the coastline file:

1. Select File / *Open*.
2. Select the file *shin.cst* and click the *Open* button.
3. Because you will be working with *ADCIRC* meshes, select the *ADCIRC* button as the coverage you want to create.

Coastline files include lists of two-dimensional polylines that may be closed or open. The open polylines are converted to *Feature Arcs* and are interpreted as open sections of coastline. Closed polylines are converted to arcs and are assigned the attributes of islands.

13.2.1 Defining the Domain


We need to define the region to be modeled. To do this:

1. Make sure you are in the *Map*  module.
2. Choose the *Select Feature Arc*  tool from the *Toolbox* and click on the coastline arc to select it.
3. Choose *Feature Objects / Define Domain*.
4. Select the *Semi-circular* option and push OK.

A semi-circular arc is created to define the region.

13.2.2 Assigning Boundary Types

Boundary types for the initial mesh generation are specified in the *Map* module. Boundary types are prescribed by setting attributes to Feature Arcs. To set the boundary types:

1. Choose the *Select Feature Arc*  tool from the *Toolbox*.
2. Double click the arc representing the ocean boundary, shown in Figure 13-1.
3. In the *ADCIRC Arc Atts* dialog, assign this arc to be of type *Ocean*.
4. Click the *OK* button to close the dialog.
5. Repeat this process for the coastline arc, but make sure the boundary type is set to *Mainland*, as shown in Figure 13-1.

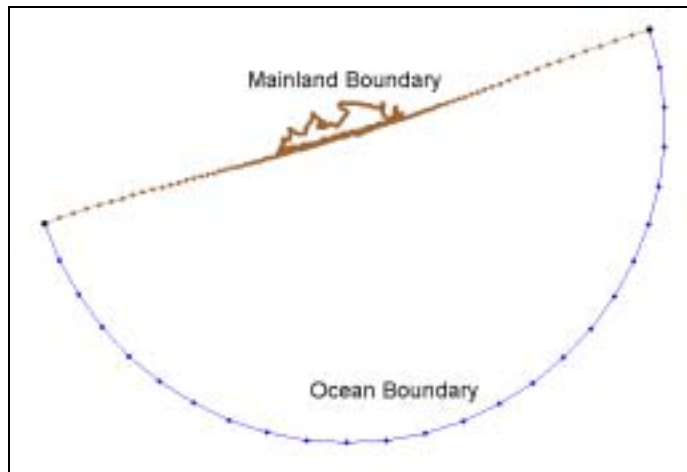


Figure 13-1 Feature Arcs after boundary types have been assigned.

13.3 Editing the Coastline File

Now that the coastline file has been read in, several modifications must be made to the data before the SHOALS file is read in.

13.3.1 Coordinate Conversions

The current coordinates of the coastline file are in a latitude/longitude geographic coordinate system. However, in order to work through this particular tutorial, we need to convert the coordinates to UTM. This will aid you in making calculations and will help preserve the accuracy of the mesh you will build later in the lesson. To convert the coordinates:

1. Choose *Edit / Coordinate Conversions*.
2. In the *Coordinate Conversion* dialog that appears, click the *Current Options...* button to define the coordinate system the data is currently in.
3. In the *Coordinates* dialog, change the coordinates for the *Horizontal System* to *Geographic NAD 27 (US)*.
4. Since the bathymetric data you will be reading in is in meters, set the *Units* for the *Vertical System* to *Meters*.
5. Click the *OK* button to close the *Coordinates* dialog.

6. In the *Coordinate Conversion* dialog, in the *Convert to* section of the dialog, set the *Horizontal System* to *UTM NAD 27 (US)*. Make sure the *UTM Zone* is set to *Zone 18 78W to 72W*.
7. Change the *Units* for both the *Horizontal* and *Vertical Systems* to *Meters*.
8. Click the *Convert* button to exit the dialog.

The coastline data has now been converted from Geographic coordinates to UTM coordinates. The coastline should now look skewed from the original coastline.

13.4 Reading in a SHOALS File

You will now read in a SHOALS file, *shin.pts*, which contains data at various locations along the coastline and throughout the region you are modeling.

1. Choose *File / Open*.
2. Select the file *shin.pts*.

The *File Import Wizard* dialog will open, allowing you to specify how the data will be read into *SMS*. For *Step 1* of the dialog, the first line in the *File preview* box is the file header. The next line shows the name of each respective column of data. In this case, the file has three data columns. The first column is the *X* Coordinate, the second column is the *Y* Coordinate, and the third column is the *depth/bathymetry*.

- Click the *Next >* button to move on to *Step 2* of the *File Import Wizard*.

The second step of the *File Import Wizard* allows you to change other specifications as you read in the SHOALS file.

- Leave the defaults and click the *Next >* button to move on to *Step 3* of the *File Import Wizard*.

The third step of the *File Import Wizard* allows you to make a coordinate conversion from the data in the SHOALS file to the current coordinate system already specified in *SMS*. Notice the current coordinate system at the top of the dialog. The data in the SHOALS file is also in UTM coordinates, so you will not need to make a conversion.

- Click *Finish* to complete the process.

Figure 13-2 shows the plot of the points read in from the *shin.pts* file.

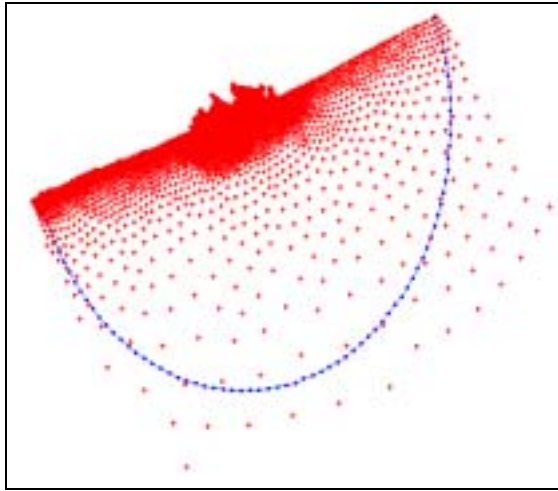



Figure 13-2 Display of shin.pts.

13.5 Shallow Wavelength Functions

The next step before you build your finite-element mesh is to create several functions for creating the finite element mesh. For this tutorial, the mesh will be generated according to the wavelength at each node. Large elements will be created in regions of long wavelengths. Conversely, smaller elements are needed closer to shore to correctly model the smaller wavelengths.

To create this shallow wavelength function from the bathymetric data:

1. In the *Scatter*  module, select *Data / Create Data Sets*.
2. In the *Create Data Sets* dialog, select the *All Off* button to turn off all of the functions.
3. Turn on the *Coastal* function option.
4. Turn on the option for creating a *Shallow Wavelength/Celerity* function.
5. Click the *OK* button to create the functions and close the dialog.

Two functions are created: celerity and wavelength at each node using shallow water wavelength equation. The celerity is calculated as:

- $Celerity = (Gravity * Nodal\ Elevation)^2$.

The wavelength is calculated as:





- $Wavelength = Period * Celerity$.

13.6 Creating Size Functions

Now that you have created the shallow wavelength function, you must make a few more conversions before you are ready to create your mesh. A size function is a multiple or variation of an already existing function. If you were to generate your mesh using the original wavelength function alone, you would get a decent mesh to work with, but we want a mesh whose density radiates out from a point in the inlet. This allows you to get more accurate results in the inlet where we are most concerned with the outcome of the *ADCIRC* run. Therefore, you now need to create a size function that works with the type of mesh we want. The final size function is found by using a variation of the original wavelength function. The final size function you will use was found through trial and error to give a nicely formed mesh. For this case, you will create several separate functions and then combine them into one final size function to use when meshing.

13.6.1 Finding the Central Point for the Mesh

Since the mesh will be generated in a radial fashion, the distance from a central point must be found. The first step is to locate the central point and then use the *Data Calculator* to compute the distances of all points from this center point. To do this:

1. Still in the *Scatterpoint*  module, zoom in to the area of the inlet as shown in Figure 13-3 using the *Zoom*  tool in the *Toolbox*.
2. Click on one of scatter points (Figure 13-4) in the middle of the inlet using the *Select Scatterpoints*  tool. Make note of this point's *X* and *Y* coordinates in the *Edit Window* at the top of the screen.
3. Frame the data by clicking the *Frame*  tool in the *Toolbox*.

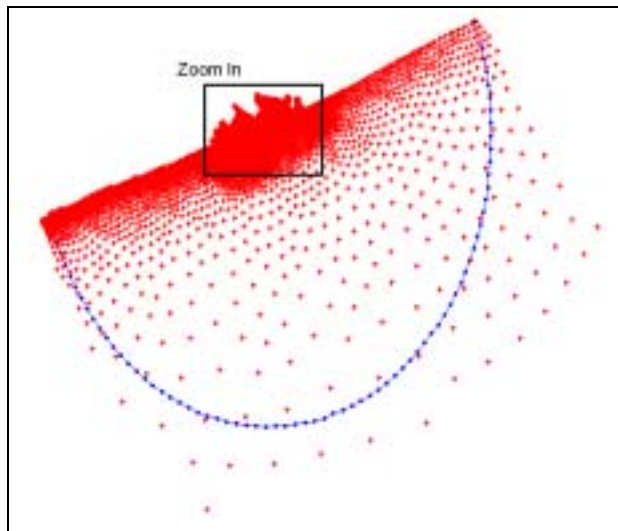


Figure 13-3 Inlet location to zoom in on.

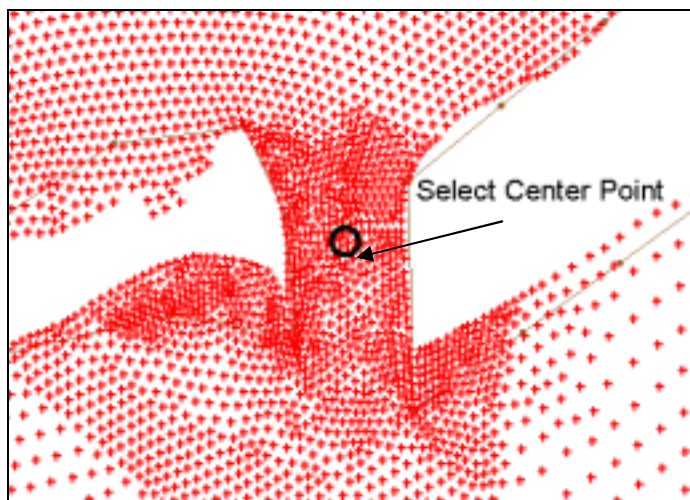



Figure 13-4 Choose a center point.

For now, turn off the scatterpoint display. However, you may turn it back on at any time during the tutorial if you so desire. To turn off the visibility of the *shin.pts* data:

1. Choose the *Display Options*  macro from the *Toolbox*.
2. In the *Display Options* dialog, under the *Scatter* tab, turn off the *Visible* option.
3. Click the *OK* button to close the dialog.

You are now ready to proceed. *SMS* has a powerful tool, called the *Data Calculator*, for computing new data sets by performing operations on scalar values and existing data sets. The *Data Calculator* will be used to create the size function.

13.6.2 Distance Function

For consistency, we will use the (x,y) location of (712768.675, 4523969.712) as the center scatterpoint for our mesh.

1. Select *Data / Data Calculator*.
2. Click the \sqrt{x} button.
3. In the *Expression* field, replace “??” using the keyboard so the expression looks like:

$$\text{sqrt}((d - 712768.675)^2 + (e - 4523969.712)^2)$$

This expression takes the x and y *locations* of each scatterpoint, which correspond to the “d” and “e” data sets respectively, and computes its distance to the point designated as the mesh center.

4. In the *Result* field, enter the name of “distance” for the data set and click the *Compute* button.

13.6.3 Initial Size Function

1. Clear the *Expression* field (delete all characters in the edit field).
2. Highlight the “Shallow_Wavelength” data set and click the *Add to Expression* button. You now should see the letter “b” in the *Expression* field at the bottom.
3. In the *Expression* field, make the equation look like “b*7”.
4. Enter the name “size” for this data set in the *Result* area and click the *Compute* button. This creates a function of 7 times the wavelength.

13.6.4 Scale Function

The last separate function before computing the final size function will be a scale factor out from the center point. It will take on the following format:

$$\text{scale} = (\text{distance}/\text{max distance})^{0.5}.$$

This scale function will range between 0 and 1, 0 being at the center point and 1 at the farthest point from the center of the mesh. This will allow the mesh to radiate out in density from the middle of the inlet. Taking the square root of the scale factor forces the elements to grow larger more quickly as one moves away from the center. To compute this function:

1. Highlight the “distance” function in the *Data Sets* window and click the *Info...* button. Notice that the *Maximum value* is 65627.258.
2. Click the *OK* button to return to the *Data Calculator* dialog.
3. Clear the equation in the *Expression* field and enter “sqrt(f / 65627.258)”.
4. Enter the name “scale” and click the *Compute* button.

13.6.5 Final Size Function

You are now ready to create the final size function that your mesh will be based on.

1. Clear the equation from the *Expression* field.
2. Click the *max(x,y)* button.
3. Replace “??,??” so the equation reads “max(50, (g*h))”. This will multiply the scale factor by the size. The minimum size of the elements will be 50 meters to prevent infinitely small elements from being created around the mesh center.
4. Enter the name “finalsize” in the *Result* field and click the *Compute* button.
5. Click *Done* to exit the *Data Calculator* dialog.

The data calculator gives you many options for building the size function. The size function created in this tutorial was created through several steps. This was done to show the many possibilities that exist for defining the size function, and ultimately for defining the finite element mesh. Other options that could be used for this or other meshes include:


- Use the wavelength multiplied by a scale factor (without using distance).
- Don’t take the square root of the scale factor for a denser mesh.
- Use a value other than 50 meters as the minimum size for a denser mesh in the channel, etc.

13.7 Creating Polygons

A *polygon* is defined by a closed loop of Feature Arcs and can consist of a single Feature Arc or multiple Feature Arcs, as long as a closed loop is formed. For initial mesh generation, polygons are a means for defining the mesh domain.


13.7.1 Building Polygons

To create polygons from the arcs on the screen:

1. Switch to the *Map*  module.
2. Make sure that no arcs are currently selected.
3. Select *Feature Objects / Build Polygons*.
4. Now a polygon has been created out of all the arcs, that can be used for the mesh.

13.7.2 Polygon Attributes

Next each polygon must be selected and have all the proper attributes set.

1. Choose the *Select Feature Polygon*  tool from the *Toolbox* and click inside the polygon.
2. Select *Feature Objects / Attributes*. (Double-clicking inside the polygon will perform this same step.) The *Polygon Attributes* dialog will open.

13.7.3 Assigning the Meshing Type

1. Select *Scalar Paving Density* as the *Mesh Type*.
2. Click the *Scatter Options...* button below the *Mesh Type*.
3. In the *Interpolation* dialog, under *Data Set*, highlight the *finalsize* function.
4. In the *Extrapolation* section, set the *Single Value* to 50.
5. Make sure the *Truncate values* option is turned on and set the *Min* to 50 and the *Max* to 5000.
6. Click the *OK* button to return to the *Polygon Attributes* dialog.

13.7.4 Assigning the Bathymetry Type

Next, the bathymetry type is selected. In this case the imported bathymetry is in the form of a scatter set.

1. Select *Scatter set* as the *Bathymetry Type*.

2. Click the *Scatter Options...* button below the *Bathymetry Type* option.
3. In the *Interpolation* dialog, make sure that *elevation* is the highlighted function under *Data Set*, leave the *Single Value* at 0.000, and make sure the *Truncate values* option is turned off.
4. Click the *OK* button.

13.7.5 Assigning the Polygon Type

1. Change the *Polygon/Type Material* to *Ocean*.
2. Click the *OK* button to close the *Polygon Attributes* dialog.

13.8 Creating the Mesh



Once the polygon attributes are set, the mesh can be generated automatically based on the options that were selected. To generate the mesh:

- Select *Feature Objects / Map -> 2D Mesh*.

Depending on the speed of your computer, this process could take anywhere from several seconds to a few minutes.

13.8.1 Mesh Display Options

After *SMS* has completed generation of the mesh, you should be able to view the bathymetry, nodes, and elements. To set the display:

1. Switch to the *Mesh*  module.
2. Select *Display / Display Options...* or select the  macro from the *Toolbox*.
3. Turn both the *Nodes* and *Contours* off and the *Elements* on under the *2D Mesh* tab.
4. Click the *OK* button to close the *Mesh Display Options* dialog.

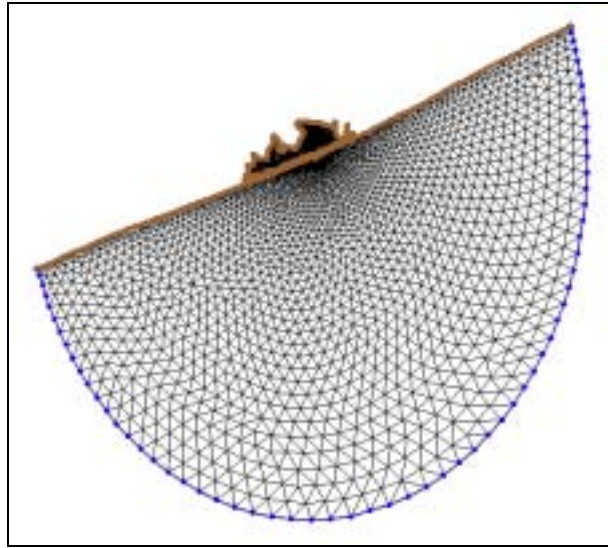



Figure 13-5 View of elements after automatic mesh generation.

Figure 13-5 shows the final mesh. Notice how the elements are smaller closer to the coast and within the inlet. Once the mesh has been created and refined, final preparations must be done in order to run *ADCIRC*. These items are renumbering of the mesh nodes and saving the grid.

13.8.2 Minimizing Mesh Bandwidth

Before running *ADCIRC*, the mesh nodes must be renumbered to minimize the bandwidth of the mesh. This allows the *ADCIRC* model to run efficiently. To do this:

1. Select the *Select Nodestring*  tool from the *Toolbox* and select the nodestring along the ocean boundary.
2. Choose *Nodestrings / Renumber*.
3. Make sure the *Bandwidth* option is selected and click the *OK* button.

The nodes have now been renumbered for the entire mesh starting with those along the ocean boundary.

13.9 Building the *ADCIRC* Control File

The control file specifies values corresponding to different parameters for *ADCIRC* runs. These parameters include specifications for tidal forcing, selection of terms to include, hot start options, model timing, numerical settings, and output control. In

order for *ADCIRC* to run properly, the mesh must be converted to Latitude/Longitude coordinates.

13.9.1 Converting Back to Lat/Lon

The model control expects the coordinates to be in latitude/longitude. The initial conversion was made to UTM coordinates for the meshing. The size function was calculated in meters, so the mesh could not be created while the coordinates were in degrees without performing more conversions (i.e. degrees \leftrightarrow meters). To convert back to Geographic coordinates:

1. Select *Edit / Coordinate Conversions*.
2. In the *Convert to* section, switch the *Horizontal System* to *Geographic NAD 27 (US)*.
3. Make sure the *Units* for the *Vertical System* are in *Meters*.
4. Click the *Convert* button.

13.9.2 Main Model Control Screen

To set up the model control for *ADCIRC*:

1. Select *ADCIRC / Model Control*.
2. Turn on the following options under *Terms* in the center of the dialog: *Time Derivative Terms On*, *Advective Terms On*, *Finite Amplitude Terms On*, and *Wetting/Drying*.
3. Click the *Options...* button beside the *Wetting/Drying* option.
4. Make sure the following values are entered in the *Wetting/Drying Parameters* dialog:

<i>Minimum Water Depth</i>	0.05
<i>Minimum # of Dry Timesteps</i>	12
<i>Number of Rewetting Timesteps</i>	12
<i>Minimum Velocity for Wetting</i>	0.02
5. Click *OK* to return to the *Model Control* dialog.
6. Locate the *Model Center* by clicking the *Find Center* button (bottom right).
7. Enter a value of 3.0 for the *Lateral Viscosity*.

13.9.3 Time Control

Next, values for the *Time Control* must be set. To set these values:

1. Click the *Time Control* button (top right).
2. Set the following values:

<i>Time Step:</i>	4.0 seconds
<i>Run Time:</i>	0.1 days (2.4 hours)

ADCIRC will generate two global output files, water-surface elevation and velocity. To set the time for the two files:

Select the *Constituent* and scroll down until *Global Elevation* is highlighted.

1. Enter the following values:
2. *Start Day* 0.0
3. *Output Every* 60 *Time Steps*
4. *End Day* 0.1
5. Change the *Constituent* to *Global Velocity* and enter the same values.
6. Click the *OK* button to return to the *Model Control* dialog.

13.9.4 Tidal Forces

For this run of *ADCIRC*, tidal forcing will be used. To define the tidal constituents that *ADCIRC* will apply at the ocean boundaries:

13.9.4.1 Tidal Potential Constituents

1. Click the *Tidal Forces* button from the main *Model Control* dialog.
2. To tell *ADCIRC* to run with tides, change the *Tidal Potential* to *On*.
3. Click the *New* button under *Tidal Potential Constituents*.
4. In the *New Constituent* dialog, make sure the *LeProvost* constituent database is selected. For this tutorial, you will be using the M2, N2, S2, O1, and K1 constituents.
5. Set the *Starting Day* as 0.00 hours on February 1, 2000 (0.00 hour, 1 day, 2 month, 2000 year). This is the date from which the tides will start.

6. Click the *M2* constituent from the *Constituents* box on the right and click the *OK* button to return to the *Tidal Functions* dialog.
7. Repeat steps 3 through 7 for the *N2*, *S2*, *O1*, and *K1* constituents.

13.9.4.2 Tidal Forcing Frequencies Constituents

Now, we have to tell *ADCIRC* to use these same constituents as forcing constituents on the open boundary of our finite-element mesh.

1. Press the *Copy Potential Constituents* button. If a prompt appears that *SMS* cannot find *m2.legi*, push *OK* and find *m2.legi* in the file browser.

SMS takes each constituent and extracts the values it needs from the *LeProvost* constituent database, placing it into the *Tidal Forcing Frequencies Constituents* box on the right.

2. Click the *OK* button twice to exit the *ADCIRC Model Control* dialog.

13.9.5 Saving The Mesh and Control Files


To save the mesh and control files:

1. Select *File / Save Project*.
2. Enter the name *shinfinal.spr* and click the *Save* button.

13.10 Running *ADCIRC*

You are now ready to run *ADCIRC*. Presently, *ADCIRC* uses a specific naming convention for its input and output files. Therefore, before *ADCIRC* can start, the basic input files must be present in the working directory, which *SMS* does automatically. *SMS* makes a copy of the active mesh file and names it *fort.14*, then makes a copy of the model control information file and names it *fort.15*. The *ADCIRC* executable also needs to be located in the directory where the files are located.

To run *ADCIRC*:

1. Move or copy the *ADCIRC* executable into the *tutorial\tut13* directory.
2. Select *ADCIRC / Run ADCIRC*.
3. If the name of the *ADCIRC* executable does not appear, click the folder icon , locate the *ADCIRC* executable, and click *OK*.

ADCIRC will run in a DOS window for 2160 timesteps. This will take just over 8 minutes on a Pentium 3, at 800 Mhz

Once the *ADCIRC* run has completed, there will be several new created. SMS copied the *shinfinal.grd* file (the mesh file saved when the project file was saved) to *fort.14* and *shinfinal.ctl* file to *fort.15*, the filenames needed by *ADCIRC*. *ADCIRC* created the *fort.63* (global elevation) and the *fort.64* (global velocity) files. There are a couple of other files that hold basic output information, but we will only focus on the elevation and velocity files for the remainder of this tutorial.

13.11 Importing *ADCIRC* Global Output Files

Each output file from *ADCIRC* is imported into *SMS* as a “Dataset.” There are two types of datasets, scalar and vector. The global elevation file is an example of a scalar dataset, while the global velocity file is a vector dataset. You will first import the global elevation file. To do this:

1. Select *Data / Data Browser*.
2. In the *Data Browser* dialog, click the *Import...* button.
3. Change the *Files of type:* to *ADCIRC Unit 63 (*.63;*.sol)*.
4. Select *fort.63* and click the *Open* button.
5. In the *Dataset File Options* dialog, select the *Add to Solution Set* option and click the *OK* button.

SMS now reads the file and adds “water surface elevation (63)” in the *Scalar Data Sets* windows under the *Generic Dataset* solution.

6. Repeat items 2 through 4, this time selecting the file *fort.64*. Changing the *Files of type:* to *ADCIRC Unit 64 (*.64;*.sol)*.

Notice that *SMS* adds “velocity (64)” to the *Vector Data Sets* window. Additionally, *SMS* has created a new scalar dataset named “magnitude (64)” that contains the magnitude of the current velocity and has placed it into the *Scalar Data Sets* window.

7. Click *Done* to exit the *Data Browser* dialog box.

13.12 Viewing *ADCIRC* Output


Once both *ADCIRC* output files have been imported, the user must decide on how to view the data. Look at the *Edit Window* at the top of the screen. This shows the

presently selected solution and data sets. Notice that the *Scalar* dataset is currently *magnitude (64)*.

- Change the *Scalar* dataset to “water surface elevation (63)”.

13.12.1 Scalar Dataset Options

A good way to view the output is to edit the contour display options. To change the contour properties:

1. Select the *Display Options*  macro in the *Toolbox*.
2. Turn off the *Elements* and *Nodestrings* display.
3. Turn the *Contours* display on.
4. Under the *Contours* tab, change the *Contour Method* to *Color contours*.
5. For the *Number of intervals*, enter 25.
6. Turn on the *Specify a range* option.
7. Enter -0.60 for the *Minimum value* and 0.10 for the *Maximum value*.
8. Make sure both the *Fill below* and *Fill above* boxes are checked.
9. Click *OK* time to exit the dialog box, and *SMS* will redraw the screen similar to Figure 13-6. Note (It will look about the same for the *Time step* 480.)

To view the data at different time steps, change the value of the presently displayed *Time Step* in the *Edit Window* at the top of the screen. The *Time Step* value is in seconds from the start of the *ADCIRC* run.

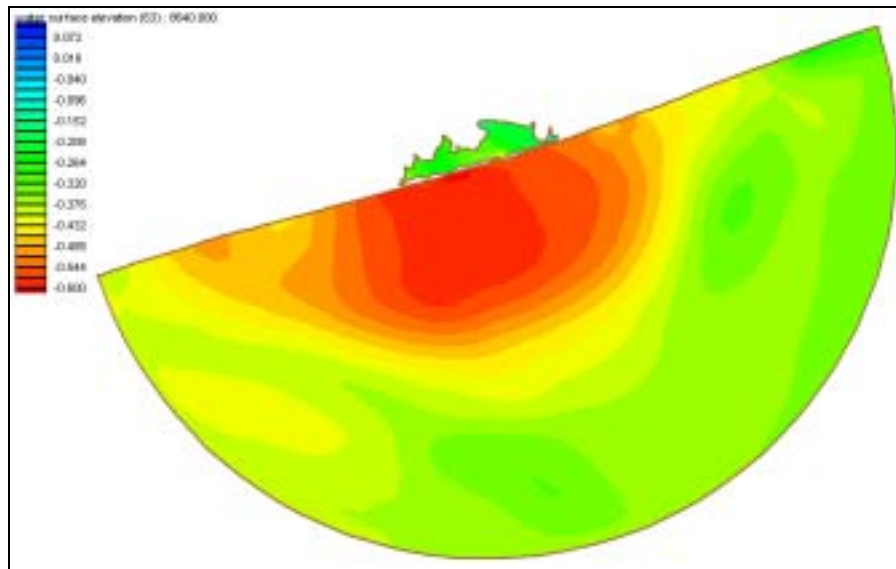




Figure 13-6 ADCIRC output from the "fort.63" file.

13.12.2 Vector Dataset Options

You can display velocity vectors several different ways. We will first view them displayed at each node, and then on a normalized grid.

13.12.2.1 Vectors at Each Node

1. Using the *Zoom* tool , zoom in on the mesh so only the bay area is visible.
2. Select the *Display Options*  macro in the *Toolbox*.
3. In the *Display Options* dialog under the *2D Mesh* tab, turn on the *Vectors* toggle.
4. Under the *Vector* tab, under the *Shaft Length* select *Define min and max length*.
5. Change the *Min length* to 10 *pix* and click the *OK* button.
6. Click the *OK* button to exit.

The screen should now look similar to that shown in Figure 13-7. You can now visualize the flow at each node through Shinnecock Bay at this particular time step.

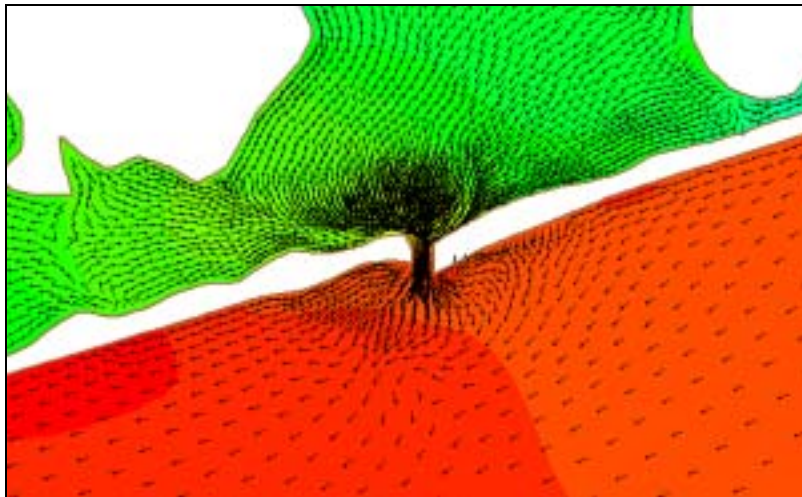



Figure 13-7 View of velocity vectors at each node.

13.12.2.2 Vectors on a Normalized Grid

1. Since the Velocity Vectors are already active, click the *Vector Options* tool  on the *Toolbox*.
2. Change the *Arrow Placement* to the *Display vectors on a Grid* option.
3. For both the *x pix* and *y pix*, enter a value of 15 and click the *OK* button.

An example of equally spaced vectors is shown in Figure 13-8. This method of displaying vectors is useful when displaying areas with both coarse and refined areas.

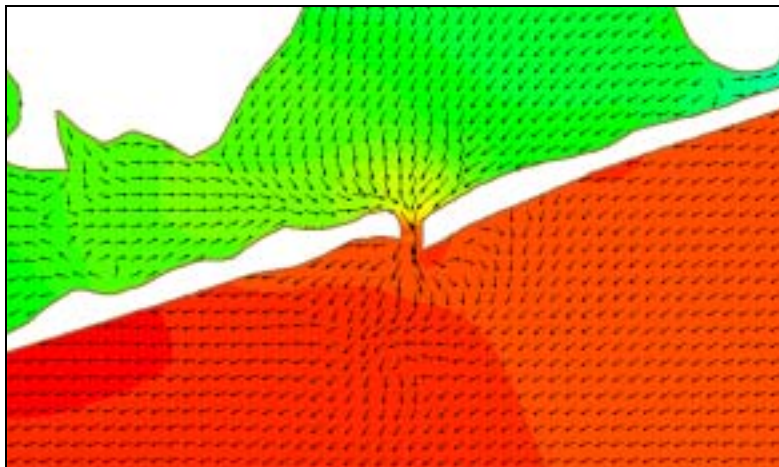





Figure 13-8 View of velocity vectors on a normalized grid.

13.13 Film Loop Visualization

Once the water-surface elevation contours and the velocity vectors are set, animations can be generated and saved. *SMS* enables the user to generate and save animations by using the Film Loop. To create a film loop of the ADCIRC analysis:

1. Select *Data / Film Loop*.
2. In the *Film Loop* dialog, click the *Next>* button.
3. Turn on both the *Scalar* and *Vector data sets* under the *Data Options* and click the *Next>* button, then the *Finish* button. (Note this will take several minutes to run.)

SMS now starts the film loop, adding one frame at a time. Once the last frame has been added to the loop, you can view the animation. To do so:

1. Click the *Play*  button.
2. You can have the animation start over again when it reaches the last time step by pressing the  button or you can have it go in reverse when it reaches the last time step by pressing the  button.

You may continue to experiment with the film loop features if you desire. Click the *Close* button when finished. You may save the film loop if you want to.

13.14 Conclusion

This concludes the *ADCIRC* Analysis tutorial. You may continue to experiment with the *SMS* interface or you may quit the program. To quit *SMS* at this point:

- Select *File / Exit*. If prompted to confirm, click the *Yes* button.

STWAVE Analysis

14.1 Introduction

This workshop gives a brief introduction to the *STWAVE* modules. Data from the Shinnecock Inlet, Long Island, New York, has been set up as an example. This example will use the mesh generated in the previous tutorial for ADCIRC. An *STWAVE* grid will be created over a small section of the *ADCIRC* mesh.

14.2 Converting ADCIRC to Scatter

14.2.1 Reading in the ADCIRC files

First, open the mesh and solution files generated in lesson 13. The files are also supplied in the *tutorial\tut14* directory. To open the files:



1. Select *File / Open...* and select *shinfinal.grd*. Push *Open* to read in the mesh file.
2. Open *fort.64* in the same manner as step 1.



The mesh will be drawn on the screen after the first step. To see the velocity vectors from the solution file:

1. Select *Display / Display Options* or the  macro in the *Toolbox*.





2. Turn on the *Vectors* toggle under the *2D Mesh* tab and push *OK*.

14.2.2 Converting to Scatter

The mesh file was read into the *Mesh* module . We need to convert the data into scattered data points (which are accessed in the *Scatter* module ). Scattered data points are used for interpolating between meshes and Cartesian grids. To convert the data to scatter points.

1. Still in the *Mesh* module , go to *Data / Mesh -> Scatterpoint*, leave the name as *scatter*, and push *OK*.
2. We won't be using the mesh anymore so turn the display off by going to *Display / Display Options* , push the *All off* button under the *2D Mesh* tab, and push *OK*.

Next, we can delete the unnecessary scatter points for speed and memory purposes:

1. Switch to the *Scatter* module . Zoom  in on the dense part as shown in Figure 14-1. Zoom by dragging a box around a region or single clicking. Hold the *Shift* key while clicking to zoom out.
2. Select the *Select Scatterpoints*  tool and choose *Edit / Select With Poly*. This allows you to click on the screen and draw out a polygon. Any points inside the polygon will be selected. Select the points around the box shown in Figure 14-1 with about a 30% margin. Double-click on the screen to close the polygon and select the points.
3. Go to *Scatter / Split Scatter Set*. Set the name to *Grid* when prompted and push *OK*.
4. Push the *Frame*  button to show both scatter sets.
5. We no longer need the larger scatter set so go to *Scatter / Delete Scatter Set*. Select *scatter* and click the *Delete* button near the bottom, and push *Done*.

Note: This is one method of reducing a scatter set. We could have also selected the points that we didn't want and deleted them.

We now have a scatter set that covers the area of interest. We can use the set to interpolate to the Cartesian grid that we will create next.

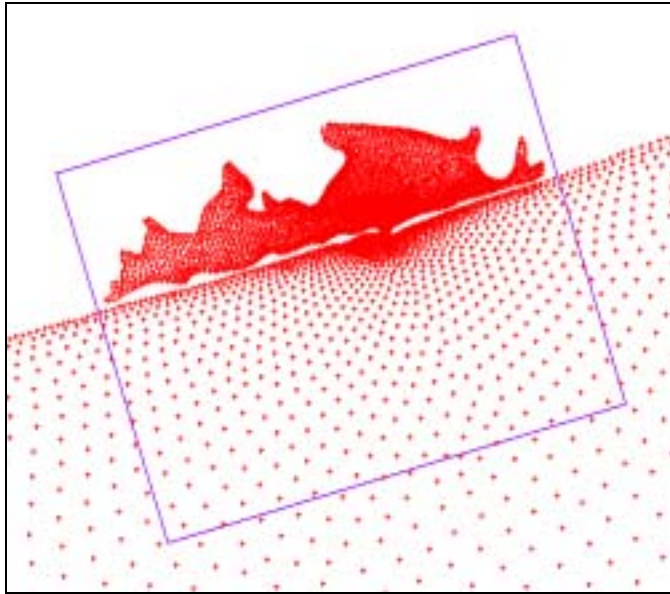




Figure 14-1. Zoomed in view of the scatter data set.



14.3 Creating the Cartesian Grid


We will now create a Cartesian grid for running *STWAVE*. The grid frame is created in the *Map* module . The *Map* module contains tools for creating GIS objects such as points, arcs, and polygons. It is also used for creating a frame, which will be filled in by a Cartesian grid.

14.3.1 Creating the Cartesian Grid Frame

To create the grid frame:





1. Switch to the *Map* module .
2. Go to *Feature Objects / Coverages...* and switch the *Coverage Type* to *STWAVE*. Push *OK*.
3. Go to *Feature Objects / Grid Frame* and push the *New Grid* button.
4. Drag and resize the grid frame by dragging the corners or edges until the grid frame fits over the desired area. Dragging a corner resizes the frame. Dragging an edge moves the entire frame.
5. Rotate the grid by dragging the circle at the bottom right corner of the frame. **IMPORTANT:** Rotate the frame until the circle is at the top right of the screen. The origin of the grid frame is now the frame corner at the bottom right of the screen.

Note: The *Frame*  and *Refresh*  tools are available while the grid frame dialog is open. It may be necessary to make the grid frame smaller and then push *OK* to exit the dialog. You can zoom in on the desired area and then position the grid frame by reopening the grid frame dialog with *Feature Objects / Grid Frame*.

6. Push *OK* when you are done positioning the frame to appear as Figure 14-1.
7. *Frame*  the data on the screen.

14.3.2 Creating the Land and Ocean Polygons

Before filling the interior of the grid frame with cells, we need to specify which cells will be ocean cells and which cells will be land cells.

1. Go back to the *Mesh*  module and choose *Data / Mesh -> Map*.
2. Select the *Mesh Boundaries -> Polygons* option and deselect the *Create New Coverage* option. This creates map polygons around the boundaries of the mesh in the active coverage. Push *OK*.
3. Switch back to the *Map*  module.
4. Select the *Create Arcs*  tool and create arcs surrounding the left part of the grid frame (outside of the polygons created in Step 1). Click to begin an arc and click to create vertices. Double click to end the arc. Connect the arcs with vertices of the ocean polygon as shown in Figure 14-2. Make sure you click on a vertex to close the loop.
5. Go to *Feature Objects / Build Polygons*. This creates polygons from closed loops of arcs.
6. Select the *Select Polygon*  tool. Double-click in the new polygon that you created (on the top) and select *Land* and push *OK*. The default for the big polygon on the bottom is *Ocean*.
7. Click somewhere on the screen outside of the polygons to deselect the polygon.

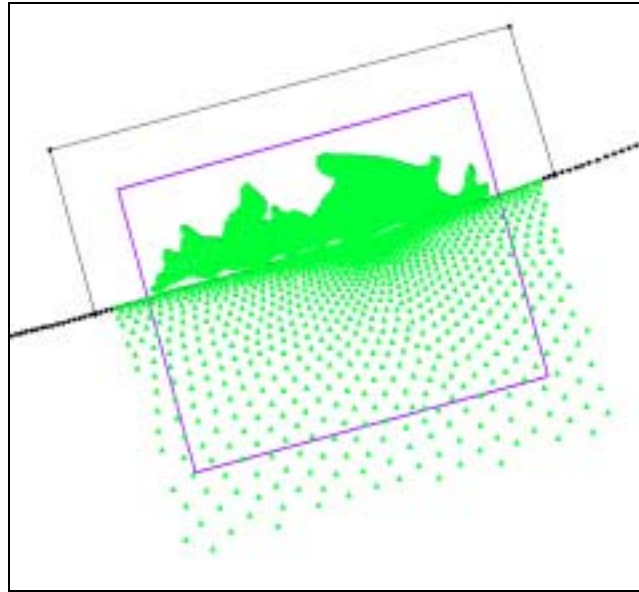


Figure 14-2. Arc created around border of grid frame.

14.3.3 Mapping to the Grid





We are now ready to fill the interior of the grid. While the grid is filling, the depth and current values will be interpolated from the scatter set and mapped to each cell. To do this:

1. Go to *Feature Objects / Map->2D Grid*.
2. In the *Map -> 2D Grid* dialog, set the *Cell Options* to *Number of Columns* and set the value to 70.
3. Notice that the *elevation* function will be used to interpolate depth values for each cell in the *Depth* section.
4. Turn the *Current* toggle on. Make sure the *Interpolated* option is selected and click the button to the right of it.
5. In the *Interpolated* dialog, change *Single Time Steps* to *Multiple Time Steps*.
6. Enter “240” in *Start time*, “900” in *Step size* and “2940” in *End Time*. This tells *SMS* to create current data at times of 240, 1140, 2040, and 2940 seconds (four sets of values around three steps of 15 minutes each).
7. Select *OK* to exit the *Interpolated* dialog.
8. Push *OK* to create the Cartesian grid.

Note on this particular grid: Because the width of the inlet and the spacing between the bay and the ocean are so small, you may need to experiment with several different sized grids to produce one that works. This is best done by moving the grid frame around and re-sizing it. However, make sure that the grid frame bounds the area of interest. Do this until the grid most closely resembles the original mesh, with only one inlet into the bay.

Note on interpolation: When interpolating you can specify a single time step or multiple steps. Single times can come from any time in the data set or interpolated between two of these times. For multiple steps, you can specify to match the steps from the data set for a specified number of steps, or you can specify a new time step size and the number of steps. SMS will generate one set of values at the beginning time specified, and one at the end of each time step. Therefore, there will be “n+1” sets of currents if you specify “n” time steps.


A Cartesian grid has been created from the grid frame. To view the grid only:

1. Go to *Feature Objects / Delete* and say *OK* at the prompt.
2. Switch to the *Scatter* module  and select *Display / Display Options* .
3. Turn the *Visible* toggle next to Grid off and push *OK*.
4. Switch to the *Cartesian Grid* module  and select *Display / Display Options* .
5. Turn on the *Ocean Boundary* and push *OK*.

14.4 Editing the Grid and Running STWAVE

14.4.1 Generating Specular Energy Distribution

We will now generate the Spectral Energy distribution.

1. Switch to the *Cartesian Grid* module  and choose *STWAVE / Spectral Energy*.
2. Push the *Generate Spectra* button. The top left spreadsheet specifies the parameters for the spectral grid itself. The first column defines the frequencies that are covered in the spectral grid. Enter “40” for Number of Frequencies, “0.01” for the Delta, and “0.04” for the Min.
3. Push the *Generate* button. The contours show the energy distribution. Select nodes (cell corners) to view/edit their energies.

4. Push *OK* to exit the *Spectral Energy* dialog.


14.4.2 Model Control

In the Model Control, *STWAVE* inputs can be set. We will change the Wind parameters:

1. Select *STWAVE / Model Control*.
2. Change the *Source Terms* under *Other Settings* to *Propagation Only*. Be careful to not change any of the *Define Grid* parameters. If those are changed, the grid will be regenerated with the new settings and depth and current information will be lost.
3. Push *OK* to exit the dialog.

14.4.3 Selecting Monitoring Stations

The final step is to select cells to act as monitoring stations:

1. Select the *Select Cell*  tool.
2. Select a cell in the inlet. The i,j location can be seen in the bottom of the screen in the status portion of the *Edit Window* when a cell is selected. (Click on the screen to select a cell.)
3. While the cell is still selected, choose *STWAVE / Assign Cell Attributes*. Turn the *Monitoring Station* toggle on and push *OK*.
4. Repeat steps 2 and 3 and select a cell in the ocean and one in the bay to be *Monitoring* cells. You can also choose an exact i, j or x, y location by going to *Data / Find Cell*.


14.4.4 Saving the Simulation

To save the simulation:

1. Select *File | Save As*, make sure the *Save as type* is set to *Project Files*, and enter the file name *shin.spr*.
2. Push the *Save* button.

14.4.5 Running STWAVE

To run *STWAVE*:

1. Select *STWAVE / Run STWAVE*.
2. If a message such as “stwave.exe – not found” is given, click the *File Browser* button  to manually find the *STWAVE* executable.
3. Click the *OK* button to launch *STWAVE*.


A window will appear and stay up while *STWAVE* runs. When *STWAVE* is finished, the window will disappear.

14.5 Post Processing

The solution files can be opened in SMS and several visualization options may be set.

14.5.1 Visualizing the STWAVE Solution

To see the solution results:

1. Open the solution files by selecting *File / Open*, select *shin.sim*, and push *OK* at the prompt.
2. Select *Display / Display Options* . Under the *2D Mesh* tab turn the *Contours* and *Vectors* toggles on.
3. Under the *Contours* tab under the *Contour Method* select *Color fill*.
4. Under the *Vector* tab change the *Shaft Length* to *Define min and max length*.
5. Set the *Min length* to 25 and the *Max length* to 50.
6. Change the *Display vectors* under *Arrow Placement* to *on a grid*.
7. Push *OK* to exit the *Display Options* dialog.

14.5.2 Visualizing Bathymetry

The solution files are stored as an SMS solution set when they are opened. The depth is not part of the solution output from *STWAVE*; it is part of the “Generic Datasets” solution. To view the bathymetry:

- Change the Solution (in the *Edit Window* near the top of *SMS*) from “shin (STWAVE)” to “Generic Datasets”. Notice that the current *Scalar* function (again at the top) is changed to “Depth.”

14.5.3 Visualizing the Direction Field

The *STWAVE* solution files are stored in the new solution. To see the wave direction vectors:

1. Switch the *Solution* back to “shin (STWAVE)” in the *Edit Window*.
2. Change the *Vector* function to “wave.”

The vector arrows show the wave direction field.

14.5.4 Visualizing the Wave Height

To see the wave height:

1. Change the *Scalar* function to “Height.”

The contours show the wave height. Repeat step 1 for “Period” and “Direction.”

14.5.5 Visualizing the Observation (Monitoring Cells) Spectra

To view the observation spectra:

1. Go to *STWAVE / Spectral Energy*.
2. Push the *Data Browser* button and select the first scalar data set named “Node_ii_jj.” Push *Done*.
3. The contours show the observation spectra for the cell at the specified *i, j* location (which should be the cell in the inlet).
4. Use the data browser (step 2) to view the observation spectra for the other cells (the one in the ocean and the one in the bay).
5. Push *OK* to exit the *Spectral Energy* dialog.

14.6 Conclusion

This concludes the *STWAVE* Analysis tutorial. If you wish to exit *SMS* at this point:

- Choose *File / Exit*.


HEC-RAS Analysis

15.1 Introduction

HECRAS was developed by the *U.S. Army Corps of Engineers Hydrologic Engineering Center*. *HECRAS* performs a step backwater curve analysis for either steady state or transient conditions to determine water surface elevations and velocities.


15.2 Preparing the Conceptual model

The first step to creating a *HECRAS* Model is to create a conceptual model which defines the river reaches (layout and attributes), the position of cross-sections on those reaches (orientation and station values), bank locations, and material zones. The conceptual model will be used to create a network schematic inside the *1D River*

Hydraulic Module .



We will create the conceptual model from an USGS quad map as well as scattered bathymetric data. To load this information, open the files *LeithCreek.img* and *elev.sup*.

The scatter points clutter the screen, but we want to know where they are so we don't create our conceptual model outside of domain of our bathymetric data. To better see the image and still know where the extents of our points are we will turn off our scatter points and turn on the scatter boundary. To do this:

1. Select the Display Options  button.
2. Select the Scatter tab.
3. Uncheck the Points toggle box.
4. Check the Boundary toggle box.
5. Click OK.

15.2.1 Creating the Coverages

We need to create a centerline coverage for our reaches and a cross-section coverage for our cross-sections. These will form the core of our conceptual model.

6. Set the current module to be the *map*  module.
7. Select *Feature Objects | Coverages...*
8. Change the name of the default coverage to “centerline” and set its type to be *1D Hyd Centerline*.
9. Create one new coverage by clicking the *New Coverage* button .
10. Change the name of the new coverage to “cross-sections” and change its type to *1D Hyd Cross-section*.
11. Make sure that the centerline coverage is active by clicking the active box in the centerline coverage row.
12. Click the *OK* button.

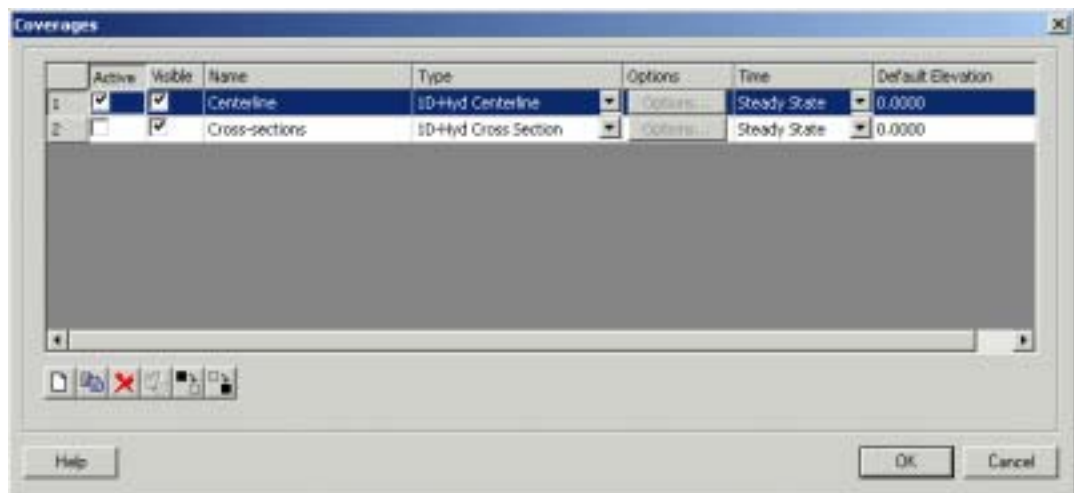


Figure 15.1 The coverage dialog.

15.2.2 Creating Centerline and Bank arcs

Before creating feature arcs we will tell SMS that we will be using the HECRAS Model. Select *Feature Objects / Set 1D Model* and choose *HECRAS*.


Centerline arcs are used to define the locations and lengths of the study reaches and assign their attributes. We will have a centerline following the main channel of Leith Creek as well as the tributary on the West. As the flows below the reservoir in the tributary on the East of the Leith Creek are small, we will disregard that reach in our simulation. To create the centerline arcs:

13. Select the *Create Feature Arc* tool .

14. Following the pattern in Figure 15.2, create the centerline of the main channel from upstream to downstream by clicking points on the centerline one at a time. Double-click the last point to indicate that it is the end of the centerline.

15. Create the arc for the west tributary upstream to downstream by clicking points on the centerline. Create the last point where the tributary meets the main channel by clicking on the main channel centerline. This splits the centerline of Leith Creek into two reaches.

This defines the centerline of the three reaches, on two rivers, that we will model in this simulation.

Bank arcs are used to define the locations of the banks and the overbank distances. The next step is to create bank arcs along both sides of each centerline arc. Using the *Create Feature Arc* tool  create new arcs where you estimate the bank locations to be (based upon contours on the background image). New arcs are assigned by default to be centerline arcs. Change them to bank arcs by following these steps:

Select the bank arcs.

16. Choose *Feature Objects / Change Arc Type*.


17. Change the type combo box to *bank arcs*.

18. Click *OK*.



Figure 15.2 Centerline and bank arc placement

The background image is no longer necessary. To turn it off:

1. Select the *Display Options*  button.
2. In the *Image* section, uncheck the *Display Image* box by clicking on it.
3. Press *OK*.

15.2.3 Naming the Centerline Arcs

Reaches are stream sections where the flowrates and other hydraulic conditions are assumed to be constant. A river can include one or more reaches, but a river is not allowed to fork in HECRAS. HECRAS has the ability to model multiple rivers (flowpaths). To assign names to our rivers and reaches:

19. Double-click the uppermost reach in the main channel.
20. Set the *River Name* to “Leith Creek”.
21. Set the *Reach Name* to “Upper Main”.

22. Click *OK*.
23. Repeat steps 1 to 4 for each reach in the map as shown below. (Note: For the Lower Main reach you can choose “Leith Creek” from the river name combo instead of typing it in.)

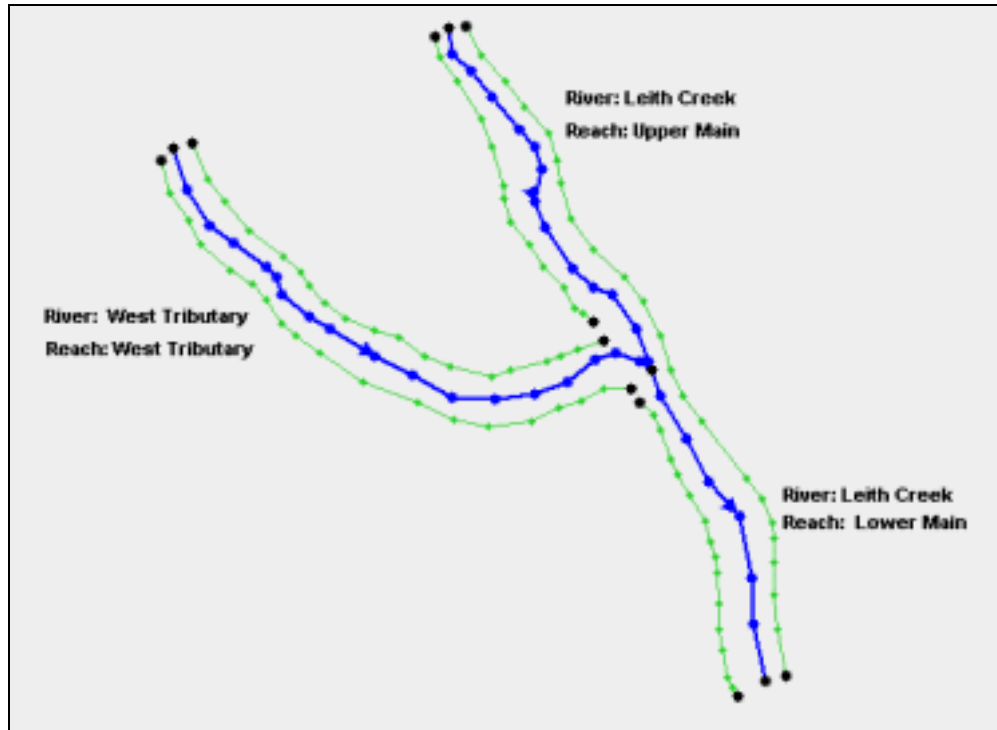



Figure 15.3 River and Reach names

15.2.4 Creating Land Use Coverage

One of the properties HECRAS uses is roughness values. We will designate materials to different areas of our model. Later we will assign each material a roughness value. The material zones are stored in SMS in a coverage of type Area Property. To load the materials data:

24. Open the file *Materials.map*.
25. Make sure the newly created Area Property coverage “materials” is active.
26. Click on the *Display Options*  button.
27. Turn on the *Polygon Fill* and *Legend* options.

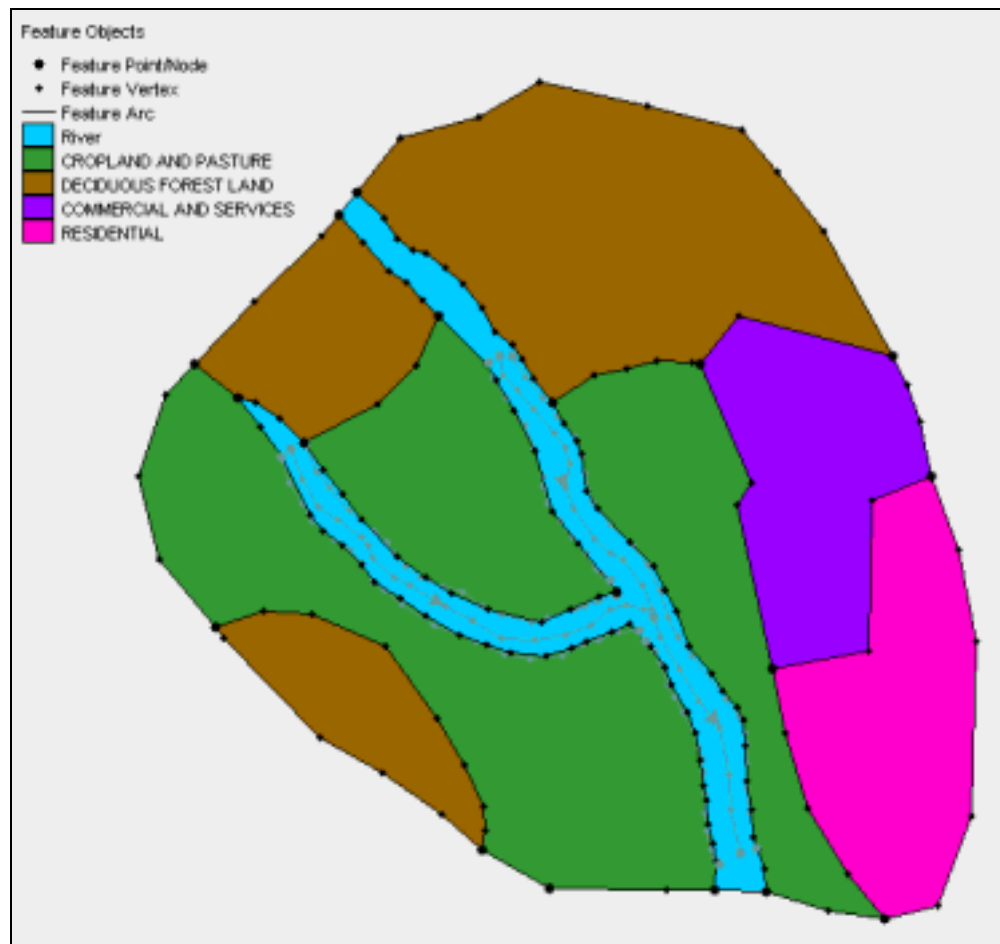


Figure 15.4 Materials used in HECRAS Simulation.

15.2.5 Creating the Cross-Sections

HECRAS associates most of its model data with cross-sections and generates solutions or output at the cross-sections. Therefore, cross-sections are the most important part of the map. *HECRAS* requires at least two cross-sections on each reach.

28. Set the current coverage in the *Coverage* combo-box on the top of the edit window to “Cross-Section.”

29. Select the Create Feature Arc tool .

30. Create at least two cross-sections on each reach by clicking a point on one side of the reach then double-clicking a point on the other side of the reach as shown in Figure 15.5.

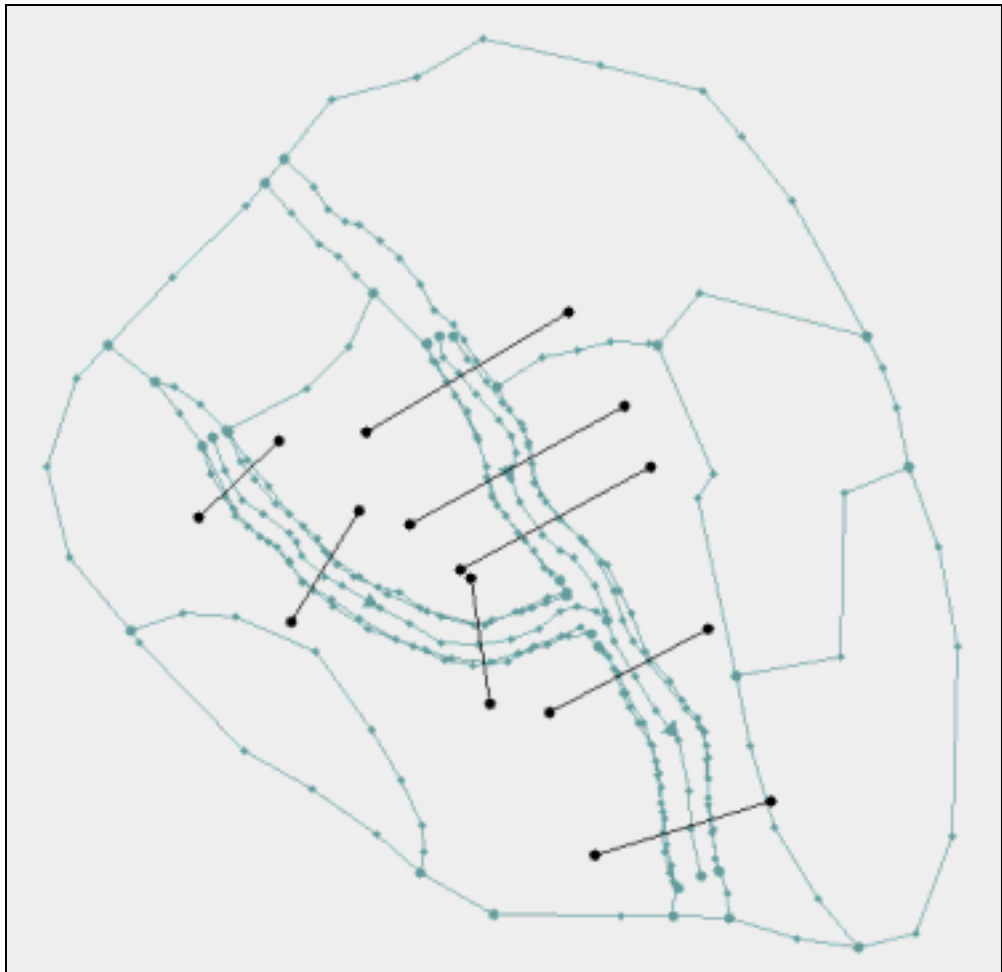



Figure 15.5 Cross-section coverage

15.2.6 Extracting Cross-Sections

In the cross-section coverage, all arcs are cross-section arcs. Their position and orientation define the location of the cross-sections in the system, but as of yet, they do not have any data assigned. We want to assign elevation data, materials, and point property locations to the cross-sections. This information will be extracted from the scattered data set (and its TIN), the area property coverage, and the centerline coverage. To extract this data:



1. Select *Feature Objects | Extract Cross Section*.
2. SMS will set the defaults to use the centerline coverage to generate point properties and to use the area property coverage “materials” to define material zones. Click *OK*.
3. SMS will prompt for a location to save the cross-section database. Enter the file name *xsecs*.

Each arc now stores a link to a cross-section database record which contains xyz data, materials properties, bank locations, and thalweg locations. To view the information at a cross-section:

31. Click on the *Select Arc Tool* .
32. Double click on any cross-section. This brings up the *River Cross Section Attributes* dialog.
33. Make sure that the reach name is assigned correctly.
34. Click on the *Assign Xsec* button. This brings up the *Assign Cross Section dialog*, which is used to view the current cross section shape and select a different cross-section from a cross-section database if desired.
35. Click on the *Edit* button. This brings up the *Cross-Section Attributes* dialog. This dialog can be used to view and/or edit the cross-section.
36. Click on the *Line Props tab* to view the materials that are assigned to the cross-section.
37. Click on the *Point Props tab* to view the locations of the left bank, right bank, and thalweg.
38. Click *Cancel* until all the dialogs are closed.

15.3 Creating the Network Schematic

SMS interacts with HECRAS using a HEC-GeoRAS geometry file. This file contains the cross-sectional data used by HECRAS in addition to three dimensional georeferencing data. To create this geometry file the conceptual model must be converted to a network schematic diagram in the *1D Hydraulic Module*. To convert the conceptual model to a network schematic:

39. Make sure you are still in the Map Module .
40. Set the current coverage in the coverage combo-box on the top of the edit window to "*Centerline*."
41. Select *Feature Objects / Map -> Schematic*.
42. Switch to the *1D River Hydraulic Module* .

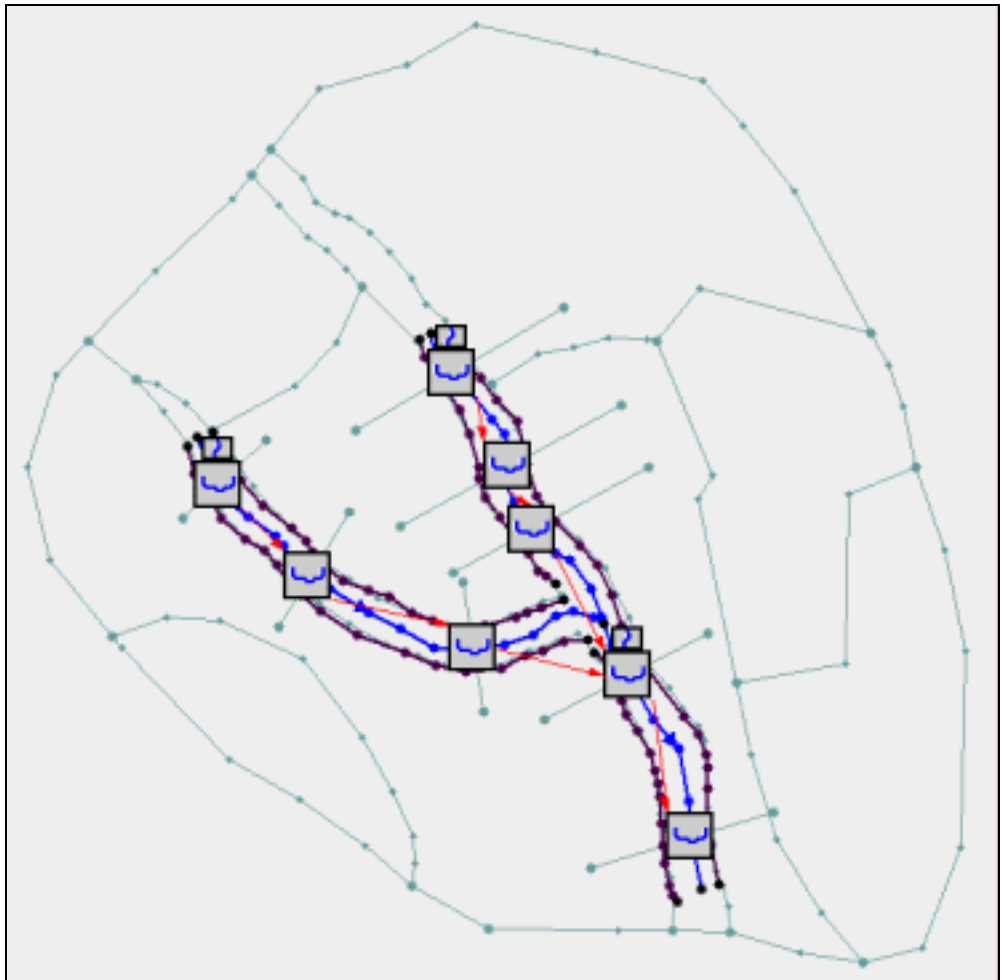


Figure 15.6 Schematic diagram

Now SMS includes two separate representations of the data. The first you created as a conceptual model which is stored as a series of coverages. The second is a numeric model stored as a schematic of cross sections organized into reaches. Modifications to the network schematic that can be used by HECRAS can be made directly in the *1D River Hydraulics Module*, or indirectly by editing the conceptual model in the *Map Module* and mapping to a new network schematic.

HECRAS needs Manning's roughness values for the materials found in the cross-section database. The roughness values are stored as part of the 1D model in the 1D River Hydraulics Module. To specify the roughness values for the each of the materials:

43. Select *HECRAS / Material Properties*.
44. Enter the roughness values for each material as shown in Figure 15.7.

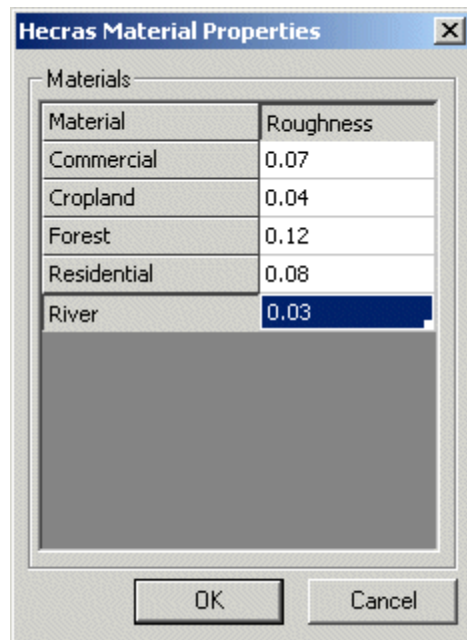


Figure 15.7 HECRAS Material Properties Dialog

45. Click *OK*.

Now we need to tell HECRAS which set of line properties in the database should be used as material types. To do this:

46. Select *HECRAS / Model Control*.

47. Select the line property name that stores the roughness values for the cross-section database. In this case, the line property is named *Materials* which came from our Area Property coverage “Materials.”

48. Click *OK*.

15.3.1 Creating the Geometry import file

Now that the simulation has been setup we need to create the geometry file. To create this file:

49. Select *File / Save As...*

50. Change Save type to *HECRAS Import File*.

51. Name the file *LeithCreekHecras.geo*.

52. Click *Save*.

15.4 Using HECRAS

In HECRAS we need to read in our geometry file, assign boundary and flow conditions, setup and run the simulation, and export the results for post-processing in SMS.

The geometry that we exported to use in HECRAS is imported through the geometry editor inside of HECRAS. To import this file:

1. Startup HECRAS.
2. Create a new project using *File / New Project*.
3. Give the project a title and a filename and click *OK*.
4. Choose *Edit / Geometric Data* to bring up the geometry editor.
5. Select *File / Import Geometric Data / GIS Format*.
6. Browse to the file LeithCreekHecras.geo.

HECRAS has a tool that will filter points that are too close together to run an analysis. To filter the points (still inside the Geometric Data editor):

1. Select *Tools / Cross Section Points Filter*.
2. Click on the *Multiple Locations* tab.
3. Choose *Leith Creek* from river combo-box.
4. The River Sta. box ought to have *All Reaches* highlighted. Click on the arrow to the right of the box to select all cross-sections.
5. Click *Filter Points on Selected XS*.
6. Click *Close*, and then *OK*.
7. Close the Geometric Data Editor by choosing *File / Exit Geometry Data Editor*.

The next step is to define the flow and boundary conditions for our reaches. To define this information:

1. Select *Edit / Steady Flow Data* from the menu.
2. For Profile 1 (PF 1) enter 4000 for the Upper Reach, enter 1000 for the West Tributary, and 5000 for the Lower Reach.
3. Click on the *Reach Boundary Conditions* button.

4. For our analysis we are going to have HECRAS compute normal depths at boundaries of our model. To do this, for each of the blank boxes in the spreadsheet, select the box and click on the *Normal Depth* button. Enter the following values for the slopes of each reach: 0.003 for the upper reach, 0.001 for the lower reach, and 0.005 for the tributary.
5. Click *OK*.
6. Click *Apply Data*.
7. Select *File / Exit Steady Flow Editor*.

We are now ready to run the steady flow analysis. We first need to set an option to set flow distribution locations so that velocity profiles will be computed. To set this option and perform the analysis:

1. Select *Run / Steady Flow Analysis* from the menu.
2. Click on *Options / Flow Distribution Locations*.
3. Change the Global subsections to 3 in each of the three fields (LOB, Channel, and ROB).
4. Click *OK*.
5. Click *Compute*. This runs the 1D analysis.
6. Close the Steady Flow Analysis Dialog.

Now that the simulation has been completed, we need to export the data for post-processing inside SMS. To export the data:

1. Choose *File / Export GIS data*.
2. Click *Browse* and choose to save the file as "*LeithCreekOut*" in your project directory.
3. Select *Export Velocity Distribution Information where available* and *Export User Defined Cross Sections*.
4. Click *Export Data*.

Exit HECRAS and go back to SMS.


15.5 Post Processing

SMS can create cross-section and profile plots of the study reach. This section will demonstrate these capabilities. With the project file still loaded, open the file

LeithCreekOut.RASexport.sdf. This will load the datasets from HECRAS into memory.

15.5.1 Profile Plots



First we will create a profile plot to see the big picture of what is happening. To create a profile plot that shows the elevation and water surface elevation of all the cross-sections along the main channel:

1. Click on the *Plot Wizard* tool .
2. Select the plot type *1D Hydraulic Profile* and click *Next*.
3. Change selected reaches to specified reaches and check the Upper and Lower Reaches.
4. Turn on the display of minimum values and turn off the display of average values.
5. Under Datasets choose specified data sets and turn on *Elevation* and *Water Surface Elevation*.
6. Click *Finish*.

The plot shows the station along the x-axis and the elevation and water surface elevation along the y-axis.

15.5.2 Cross Section Plots

We will now create a cross-section plot to look at velocity distributions at various cross-sections. To create this plot:

1. Click on the *Plot Wizard* tool .
2. Select the plot type *1D Hydraulic Cross Section* and click *Next*.
3. Leave the Cross-sections as selected cross-sections and the datasets as the active data set. Click *Finish*.
4. Using the scalar combo box in the edit window, change the dataset to *velocity*.
5. Using the *select Cross-section tool*  select a cross-section in the schematic.

Look at the velocities of various cross-sections by selecting different cross-sections.

15.5.3 Post Processing Experimentation

You can now experiment with the plot tools to visualize the other data computed by HECRAS.

15.6 Conclusion

This concludes the HEC-RAS Analysis tutorial. If you wish to exit *SMS* at this point:

- Choose *File / Exit*.

BRI-STARS Analysis

16.1 Introduction

BRI-STARS is a generalized semi-two-dimensional water and sediment-routing model. *BRI-STARS* computes water surface, velocities, as well as sediment deposition and scour. *BRI-STARS* has the additional ability to simulate channel widening/narrowing as the simulation progresses.

In this lesson a *BRI-STARS* model will be created by extracting cross sections from 2D topography. The model will then be run through the *BRI-STARS* analysis package.

16.2 Reading in the image

Using *File / Open*, load the image file *redfox.img* from the tutorial directory.

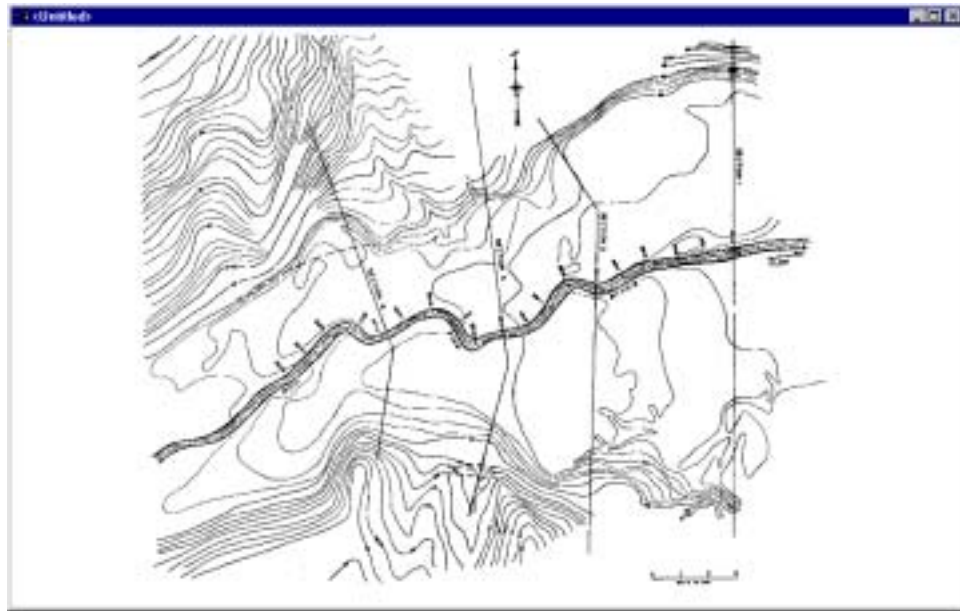



Figure 16-1 The redfox tiff image.

16.3 Creating a Conceptual Model

The *Map* module is used to create *conceptual modes* of the flow area. The conceptual model uses various *coverages*. Each coverage contains *feature points*, *feature arcs*, and *feature polygons*, which define a set of related data. See the *SMS Reference Manual* for more information on the *Map* module. The first coverage that will be created is for the centerline. To create this coverage:

1. Switch to the *Map*  Module.
2. Select *Feature Objects | Coverages*. Change the *Coverage type* to *1D-Hyd Centerline*. The *1D-Hyd Centerline* coverage is used to create feature objects related specifically to one-dimensional river models.
3. Enter the name *Centerline* for this coverage.

16.3.1 Defining a Centerline

A centerline arc is used to define the river profile. The first feature arc that is created in a *1D-Hyd Centerline* coverage by default is assigned as a centerline. To create this arc:

1. Choose the Create Feature Arcs  tool from the Toolbox.

2. Create the centerline arc by clicking out a line in the middle of the river. Centerline arcs are created from upstream to downstream (left to right). When creating the arc, you can backup by pressing the Backspace key. To abort the arc and start over, press the Esc key.
3. End the arc by double-clicking the last point.

When you are finished, you will see a centerline arc with arrows pointing downstream, as shown in Figure 16-2.

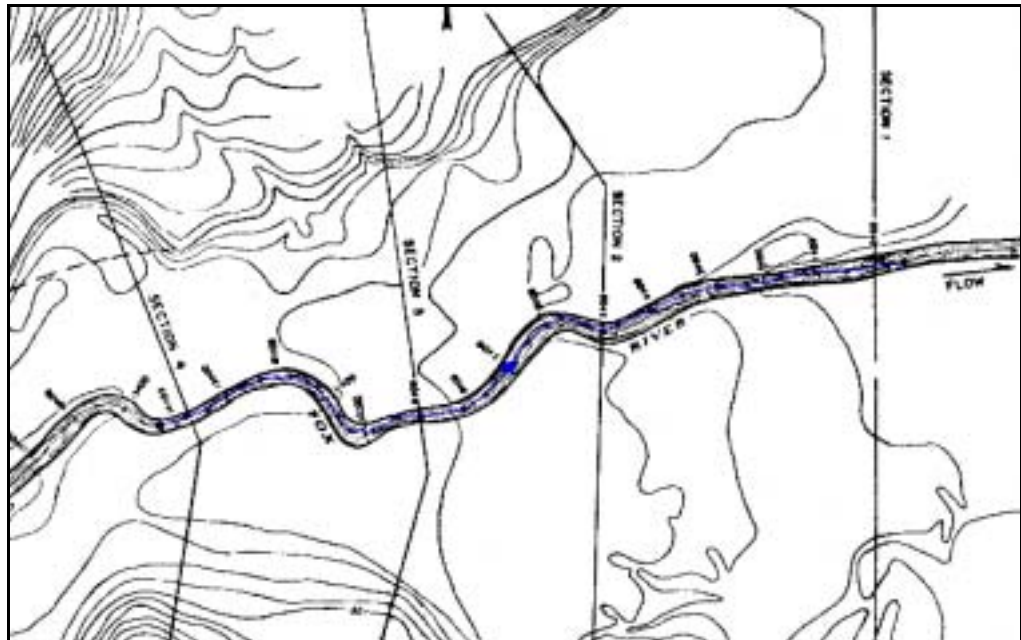



Figure 16-2 Creation of centerline


16.3.2 Creating Bank Arcs

Bank arcs can be used to tell BRI-STARS the locations of the overbanks and channel. Create two bank arcs along the Red Fox river using the same process used to create the centerline arc. The new arcs created will be generic arcs, to change them to bank arcs:


1. Select both arcs using the *Select Feature Arcs*  tool while holding the shift key.
2. Select *Feature Objects / Change Arc Type*.
3. Select *Bank Arc* from the combo box.

16.3.3 Creating Cross Sections

Cross section arcs are created inside a cross-section coverage. To create this coverage:

1. Select *Feature Objects / Coverages*.
2. Click on the New Coverage button .
3. Change the type to *1D-Hyd Cross Section*, and change the name to Cross Section Coverage.

Cross section arcs must be created across the river. *BRI-STARs* uses cross sections to compute the geometry of the river and to perform the flow analysis. Create feature arcs where they are drawn onto the image. To create the cross section arcs:

1. Choose the *Create Feature Arcs*  tool from the *Toolbox*.
2. Create an arc along *Section 1*.
3. Repeat to create the other three cross sections.

When *SMS* converts feature arcs to cross-sections, the number of vertices in the cross-section will be determined by the density of the underlying elevation data. We will read in the elevation data and extract the cross-sections later.




Figure 16-3 Map with cross-sections created.

16.3.4 Creating an Area Property Coverage

An *Area Property* coverage uses feature arcs and polygons to divide the area into zones with similar hydraulic characteristics identify by a material type. The material types we will use are: river, farmland, brush, and woodland. Later the material types will be assigned a roughness value. To create an area property coverage:

4. Select Feature Objects | Coverages.
5. Click the New button to create a new coverage and enter the name Land Use.
6. Change the Coverage type to Area Property.
7. Click the OK button to exit the Coverages dialog.

The new coverage will become active and the other coverages will dim. Material zones need to be created in this new coverage. To do this:

1. Choose the Create Feature Arcs  tool from the Toolbox.
2. Create a box around the centerline and the cross section arcs to define the land area. Be sure to enclose the entire centerline and all of the cross section arcs inside the box.
3. Create arcs to separate the different land types. Create arcs in approximately the same locations as shown in Figure 16-4.

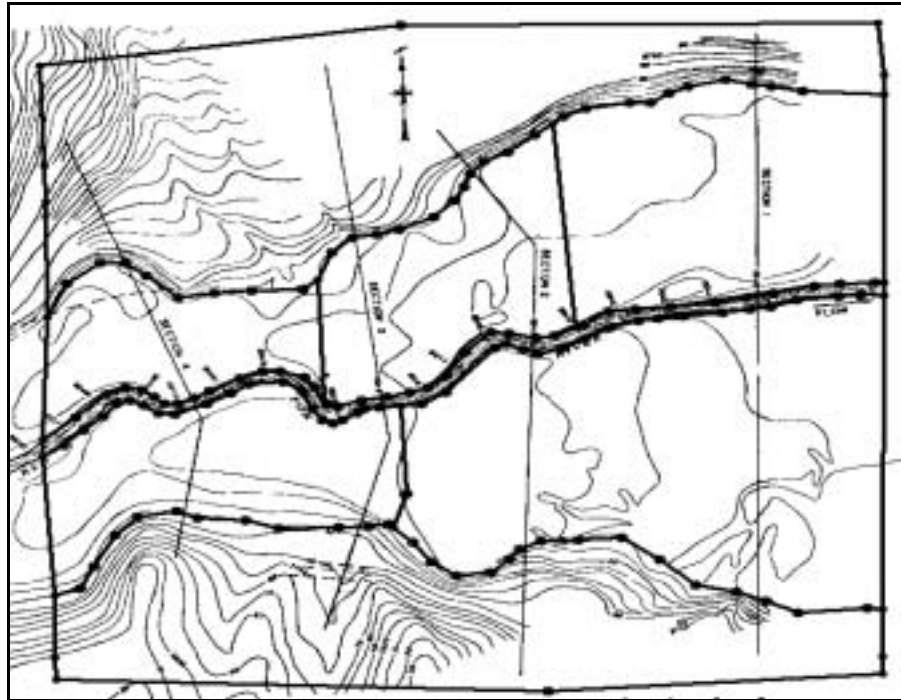



Figure 16-4 The land area boundaries.

With the arcs defined, polygons can be created to define the material zones. To create the polygons from the arcs:

1. Select Feature Objects | Clean. Accept all default values and click the OK button. This assures that polygons are ready to be built. You should always clean the feature objects before building polygons.
2. Select Feature Objects | Build Polygons. At the prompt, click the OK button to use all feature arcs.


All material zones should now be defined by polygons. These polygons can be selected using the *Select Polygon*  tool.

16.3.5 Assigning Material Types

With the polygons built, a material type can now be assigned to each polygon. First the materials must be created. To create the materials:

1. Choose the Edit | Materials Data command.
2. Click the new button 3 times.
3. Name the materials channel, brush, farm, and woods.
4. Click the OK button.

Next, to assign the material:

1. Choose the *Select Polygon*  tool from the *Toolbox*.
2. Double-click inside the polygon that includes the river channel.
3. In the *Land Poly Atts* dialog, select the *Material* option and then choose channel.
4. Click *OK*.
5. Repeat this process for each of the material zones, assigning the materials as shown in Figure 16-5.

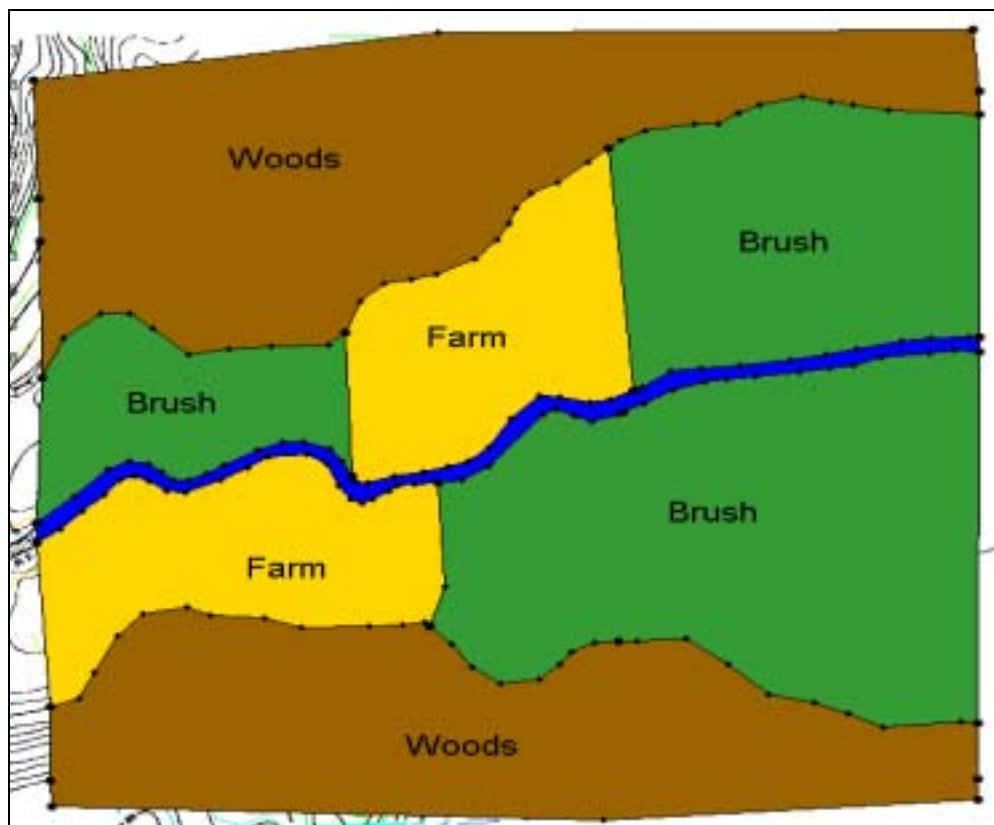


Figure 16-5 Material type boundaries.

16.3.6 Importing Topographic Data

The conceptual model defines the locations of the river centerline, the cross sections, and the material zones. However, there is no topographic information for defining elevation data. Topographic data can be taken from scattered data points, an existing finite element mesh, dxf data, or a digital elevation model. For this example, we will take the elevations from a scatter set which overlaps the model area. To open the scatterset:

1. Select File | Open.
2. Open the file redfox.sup in the redfox tutorial directory.

The display will refresh, with the triangulated network on top of the feature objects, as shown in Figure 16-6. This mesh was defined by digitizing the contour data from the tiff image. It will provide the topographic information that is required to generate cross sections for *BRI-STARs*.

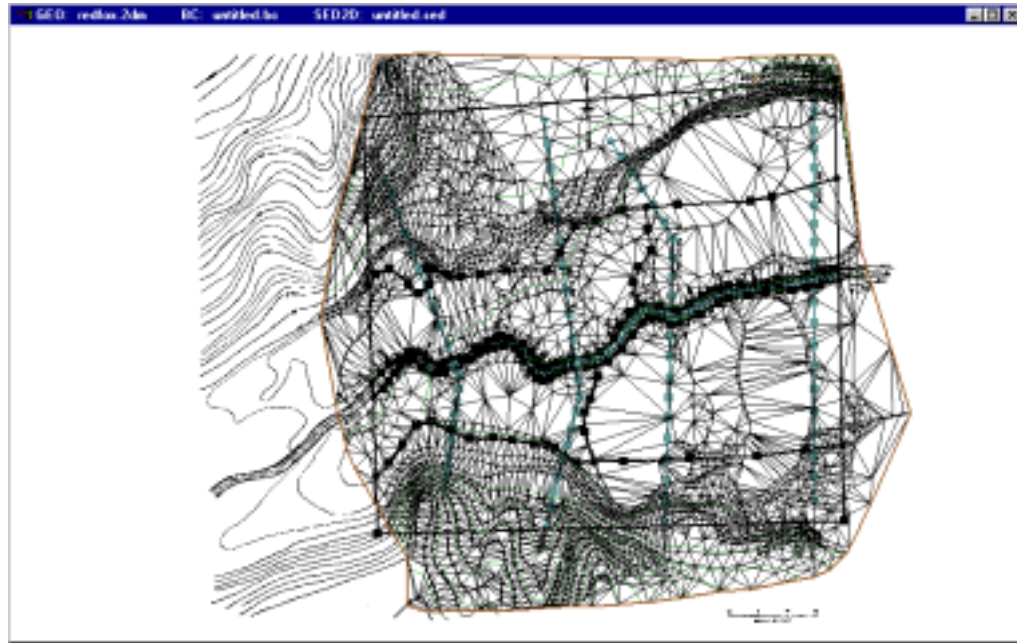



Figure 16-6 The finite element mesh on top of the map data.

16.3.7 Extracting the Cross Sections

To create a BRI-STARs Model our cross-section locations, topographic information, thalweg and bank locations, and material zones must be converted to a cross-section database. To convert our feature and scatter data to a cross-section database:


1. Switch to the *Map Module*  and the *cross-section coverage*.
2. Select *Feature Objects | Extract Cross Section* to bring up the *Extract Cross Section dialog*.
3. Make sure the options are set to use all cross-sections, to generate point properties from the centerline coverage, and to generate material zones from the land use coverage.
4. Click *OK*.

5. Enter the filename xsecs, and click Save.

We just created a cross-section database which stores all of the cross-section information. Each feature is assigned a link to a specific cross-section in this file. The main file for the cross-section database is xsecs.idx. This file may be opened inside SMS to edit the individual cross-sections. For more information about the cross-section database look under Cross-section database in the SMS Help file.

16.3.8 Assigning Reach Attributes


Now that we have the geometry and cross-section database setup for our model, we need to setup our attributes. The attributes that are assigned to a reach include the name and the boundary conditions. We have provided the inflow flowrate hydrograph, sediment inflow hydrograph, and the time varying stage values. To set the reach attributes:

1. Switch to the centerline coverage.
2. Make sure that the select arc tool is active .
3. Double click on the centerline arc.
4. Change the reach name to “Red Fox River”.
5. Click on the button *Model Atts*.
6. Change the control station type to *Lake*.
7. Click on the *Flowrate xyseries button* and import the curve “flow.xys.”
8. Similarly, for the Sediment inflow import the curve “sed_inflow.xys.”
9. Set the stage values to come from tabular data using the curve “stage.xys.”
10. Click OK in both windows.

16.3.9 Assigning Cross Section Attributes

Besides the information stored in the cross-section database, each cross-section has additional attributes that must be assigned. The only attribute we need to set is for sediment gradations for each cross-section. Sediment gradations must be assigned in order to perform sediment scour/deposition analysis. Each of the sediment gradation curves used inside SMS are stored in a list. If a curve is used in multiple locations, rather than reload the curve from disk the curve can be selected from the list of available curves. To assign the sediment curves to the cross-sections:

1. Switch to the cross-section coverage.

2. Make sure that the select arc tool is active .
3. Double click on the furthest upstream cross-section (furthest left).
4. Click on the Model Atts button.
5. Click on the *sediment gradations* xyseries button.
6. Using the import button, import the curves “sed_grad_A.xls” and sed_grad_B.xls.”
7. Select *Sediment Curve A* as the Selected Curve and click *OK* three times to exit the dialog.

Repeat this process for the rest of the cross-sections except that the curves do not need to be imported again. Simply select the curve that should be assigned to each cross-section. Assign the 2nd cross-section to also use *Sediment Curve A* and the two bottom cross-sections to use *Sediment Curve B*.


16.3.10 Saving the Data

Now would be a good time to save everything we have done up until this point. To save an SMS project file:

1. Select *File / Save Project*.
2. In the *Filename* field, enter the name *redfox1*.
3. Click the *Save* button.

16.3.11 Creating the Schematic

The conceptual model must be converted to a schematic diagram in the 1D Hydraulic Module to create the BRI-STARS simulation. To create the schematic diagram:

1. Switch to the Map Module .
2. Switch to the centerline coverage.
3. Select *Feature Objects / Map -> Schematic* from the menu.

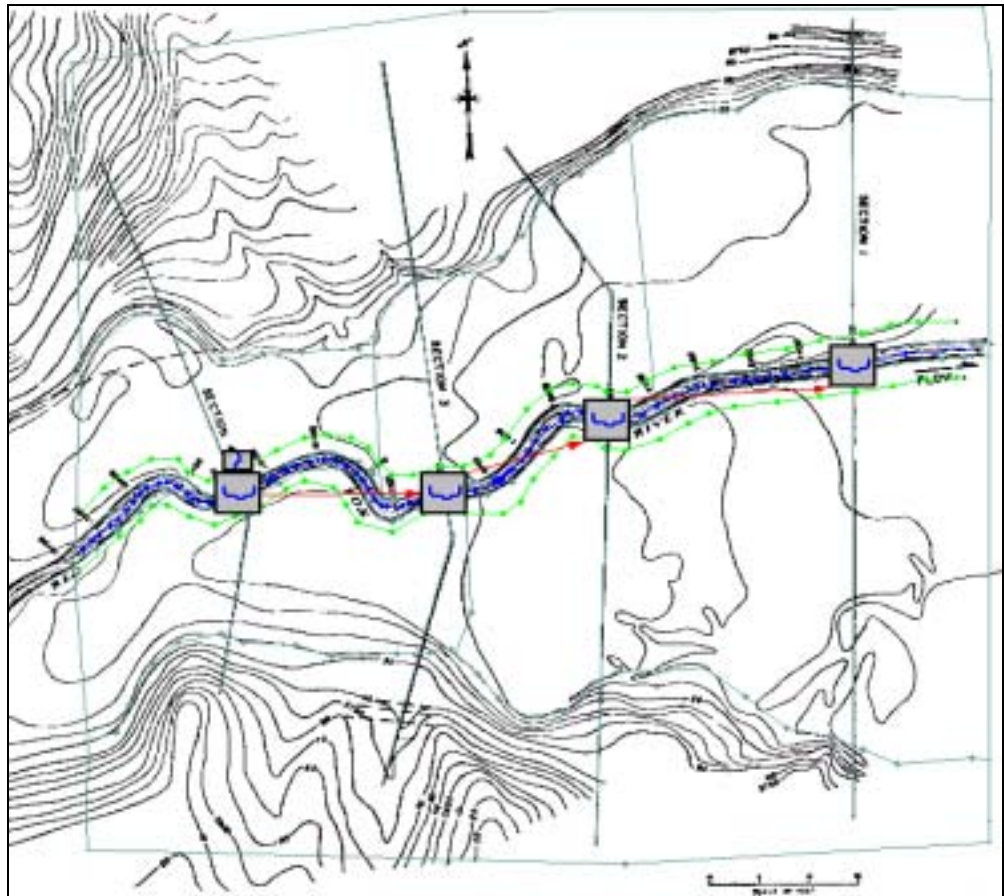



Figure 16-7 The BRI-STARS simulation with schematic diagram.

16.41D River Hydraulic Module

Within the 1D River Hydraulic Module we can set model control parameters, assign material values, edit reach attributes, and edit cross-section attributes. Since our reach and cross-section attributes were defined within the conceptual model. We only need to assign material values and set model control parameters.

16.4.1 Assigning Material Values

The material zones were assigned to the cross-sections in the database from the Area Property coverage. Although the material zone locations have been assigned to the *BRI-STARS* model, the material properties have not. To define the material properties:

1. Switch to the 1D River Module .
2. Select *BRI-STARS / Material Options*.

3. Assign the following *Manning's n* values (roughness) for the materials:
river_channel = 0.025, *woods*=0.10, *brush*=0.067, and *farm*=0.05.



16.4.2 Defining The Control Parameters

Model control parameters such as time controls, stream tubes, equations, and output controls must still be defined. To define the model control parameters:

4. Select *Bristars / Model Control* from the menu.
5. If desired you may enter titles for the simulation. These are not used by SMS or BRI-STARS.
6. Within the *Time Control* box set the following options:
 - Number of timesteps = 80.
 - Timestep size = 3 hours, 0 minutes.
 - Sediment computations per timestep = 1.
 - Water surface convergence criteria = 0.01.
 - Max iterations to reach convergence = 20.
7. Under *Stream Tubes* set the number to 3, evenly distributed.
8. Under *Misc* click the *Printout Options* button and set it to print all hydraulic computations, every 10 timesteps, from timestep 5 to timestep 100.
9. Turn on sediment transport and click on the options button to bring up the *Sediment Options* dialog.


The sediment options dialog is used to specify parameters that are used to determine scour/deposition along the study reach. These parameters include the sediment equation, active layer thickness, water temperatures, sediment size groups, and pier scour equation.


The sediment size groups are ranges of sediment sizes that are used in BRI-STARS sediment transport equations. The red lines inside the sediment size group bar represent the breakpoints between the size groups. Breakpoints can be added using

the add breakpoint tool . One breakpoint at a time may be selected with the select breakpoint tool . Selected breakpoints can be moved by dragging them with the mouse, or by changing its locations using the edit boxes above the breakpoint bar. A selected breakpoint can be removed with the backspace or delete key.

To set the sediment options:

1. Set the equation to use *Molinas and Wu – BMF*.
2. Leave the *Active Layer Thickness* as the Mean Sediment Size * 50.0.
3. Click on the button to the right of *Water Temperatures* to bring up the XYseries editor. Set the *Water temperature* xyseries to a constant 72 degrees by putting 72 degrees at time zero and 72 degrees at time 240.
4. Set sediment breakpoints at values 0.063, 0.25, 0.5, 1.0, 2.0, 4.0, and 10.0.

Breakpoints are created using the *Create Breakpoint Tool* .

Breakpoints are selected using the *Select Breakpoint Tool* . When a breakpoint is selected the edit boxes above the breakpoint bar will become enabled to allow a new number to be entered for that breakpoint. The left edit box is used for the minimum breakpoint, the right edit box is used for the maximum breakpoint, and the middle edit box is used to set all the breakpoints in between. (Hint: there are already breakpoints at the min and max).

5. The dialog should now look like Figure 16-8. Click *OK* twice to close the dialogs.

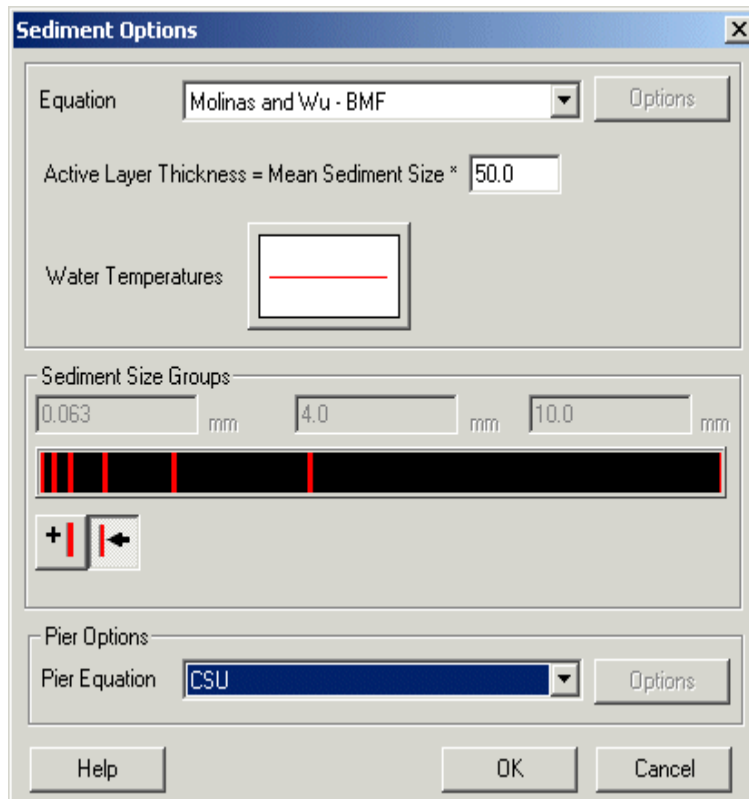


Figure 16-8 The sediment options dialog.


16.4.3 Saving the Simulation

Now that the river model has been defined, it is ready for a *BRI-STARs* analysis. Before the analysis can be run, the simulation must first be saved. To save the simulation:

1. Select *File / Save as*.
2. Save the project as *redfox.spr*.

16.5 Running BRI-STARs

To run *BRI-STARs*:

1. Select *BRI-STARs / Run BRI-STARs*. A prompt appears to show the location of the *BRI-STARs* executable.
2. If the *BRI-STARs* executable is not found, or if SMS finds an older version that you don't want to use, click the *File Browser*  button to manually select the proper *BRI-STARs* executable.
3. Click the *OK* button to run *BRI-STARs*.
4. When the *BRI-STARs* menu comes up select *File / Run* (all the options and files have all been set).

When the model finishes running, close *BRI-STARs*.

16.6 Post Processing

16.6.1 Opening the output files

BRI-STARs creates four output files. These files are:

- The *Output* file (.OUT) with model output specified in the *Printout Options* dialog inside the *Model Control* dialog.
- The *Cross Sections* file (.XSC) that contains cross-section elevations after each timestep (after scour and deposition).
- The *WSP* file (.WSP) that contains water surface and thalweg elevations output.


- The *Messages* file (.MSG) that contains convergence results, warnings, and other messages.

SMS has the ability to read information from the output file and the Cross Sections file. It is good practice to open the Messages file in a text editor to check convergence and look for warnings.

Read in the output file and the Cross sections file using *File / Open*.



16.6.2 Creating Plots

Profile plots are used to visualize changes along the study reach. Scour and deposition will mostly occur where there is abnormally high or low velocities. We will create a profile plot to see which locations scour and deposition are likely to occur. To create the profile plot:

1. Click on the Plot Wizard tool .
2. Select 1D Hydraulic Profile from the list of plot types, click Next.
3. Set the values to display to show Minimum and Maximum values.
4. Leave the dataset as active dataset and timestep the active timestep.
5. Click Finish.
6. Change the Solution to redfox.out and the scalar to velocity.

Change the timestep combo box to see what is happening at different times. Predict where scour/deposition is likely to occur.

Cross-section plots give more detailed results of what is happening at a particular station. We want to create a plot that shows the bed elevations for the first and last timesteps at selected cross-sections. To create the cross-section plot:

1. Click on the Plot Wizard tool .
2. Select 1D Hydraulic Cross Section from the list of plot types, click Next.
3. Choose specified data sets and turn on the Bed Elevations toggle.
4. Choose the specified time steps and choose 30 and 240 (hours).
5. Click Finish.
6. Using the select cross-section tool  select a cross-section to plot.

** Note: *BRI-STARs* does not compute scour or deposition at the furthest upstream cross-section so no change will appear at that location.

16.7 Conclusion

This concludes the BRI-STARs Analysis tutorial. If you wish to exit SMS at this point:

- Choose *File / Exit*.

Observation Coverage

17.1 Introduction

An important part of any computer model is the verification of results. Surface water modeling is no exception. Before using a surface water model to predict results, the model must successfully simulate observed behavior. Calibration is the process of altering model parameters until the computed solution matches observed field values within an acceptable level of accuracy. *SMS* contains a suite of tools in the *Observation Coverage* to assist in the model verification and calibration processes.

The observation coverage consists of *Observation Points* and *Observation Arcs*, which help analyze the solution for a model. Observation points can be used to verify the numerical analysis with measured field data and calibration. They can also be used to see how data changes through time. Observation arcs can be used to view the results for cross sections or river profiles. This tutorial is based on a *FESWMS* finite element model, but the calibration tools in *SMS* can be used with any model.

17.2 Opening the Data



To open the *FESWMS* simulation and solution data:

1. Select *File / Open...*

2. Open the file *observe1.spr* from the *tut17* directory. If geometry data is still open from a previous tutorial, you will be warned that the existing mesh will be deleted. If this happens, click *OK* to the prompt.

17.3 Viewing Solution Data

An initial solution has already been created with this data file and was opened with the project. When the solution file is opened into *SMS*, various scalar and vector data sets are created. By default, the active data sets are the *velocity mag* scalar data set and the *velocity* vector data set. Several display options should be changed. To do this:

1. In the *Mesh*  module select *Display / Display Options...* .
2. Click the *All off* button and then turn on the *Contours* and *Mesh boundary* options.
3. Click *OK* to exit the *Display Options* dialog.

After setting the display options, the mesh data will appear as shown in Figure 17-1.

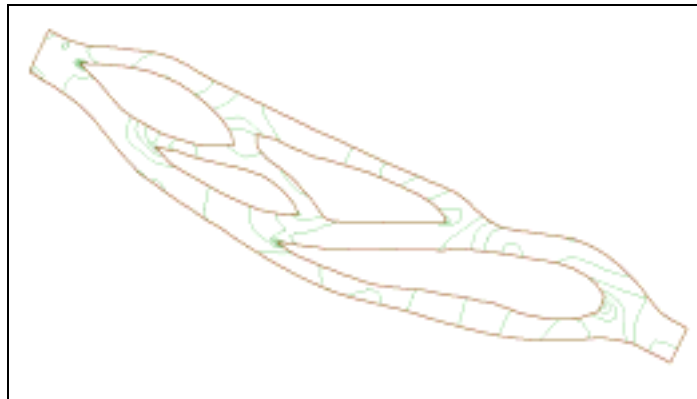




Figure 17-1 The mesh contained in *observe1.spr*

17.4 Creating an Observation Coverage

Before creating observation points and arcs, an observation coverage must exist. To create an observation coverage:

1. Switch to the *Map*  module.
2. Select *Feature Objects / Coverages....*

3. Click the *New*  button to create a new coverage.
4. Change the *Name* of the new coverage to "Observation," and change the *Type* to *Observation*
5. Click *OK* to exit the *Coverages* dialog.

At this point, an observation coverage has been created.

The *Observation Coverage* dialog can now be used to specify what data to use in calibrating the model and to edit observation points and arcs. To bring up the *Observation Coverage* dialog select *Feature Objects / Attributes...*

17.5 The Observation Coverage

In this tutorial, observation points will be used to calibrate the model; however, observation arcs or a combination of arcs and points can be used instead depending on the data collected in the field. Observation arcs work similar to observation points and any differences will be pointed out as the tutorial proceeds.

The *Observation Coverage* dialog can show the attributes for either observation points or observation arcs, but not both at the same time. The *Feature Object* combo box determines which attributes are currently being shown in the *Observation Coverage* dialog.

The upper spreadsheet is called the *Measurements* spreadsheet and the lower spreadsheet is called the *Observation Objects* spreadsheet. The titles of these spreadsheets change depending on what is selected as the feature object. Right now, the title of the *Measurements* spreadsheet is simply "Measurements" and the title of the *Observation Objects* spreadsheet is "Observation Points." Select *arcs* as the feature object and the titles of the *Measurements* and *Observation Objects* spreadsheets will change to "Flux Measurements" and "Observation Arcs," respectively.

Before continuing, it should be pointed out that observation points use single values measured in the field such as velocity and water surface elevation to calibrate the model. On the other hand, observation arcs use fluxes that have been computed across the arc to calibrate the model. Therefore, measurements for observation arcs are called "Flux Measurements."

17.5.1 Creating a Measurement

By default, when the *Observation Coverage* dialog is first opened, a *Measurement* is created. A measurement represents the solution data that is compared to the observed field data in the calibration process. For observation points, a measurement

is tied to either a scalar or a vector data set. This data set is unique to the measurement and cannot be tied to another measurement. For observation arcs, a measurement is tied to both a scalar and a vector data set. Again, this combination of data sets is unique to the measurement.

In addition to a unique *Name* and *Data Set(s)*, two other parameters are used to define the data represented by a measurement: *Trans* and *Module*. When analyzing data that varies through time, select the *Trans* toggle. The *Module* of a measurement refers to the SMS module where the computed data is stored. (The *Module* is set by default and normally does not need to be changed.)

The default measurement can be edited or a new measurement can be created. To edit the default measurement:

1. Make sure *points* is selected as the feature object.
2. Type “Velocity” as the *Name* of the measurement.
3. Select *velocity* as the *Data Set* (not *velocity mag*).

Now that a measurement has been defined, observation points can be created and edited.

17.6 Creating an Observation Point

Observation points are created at locations in the model where the velocity or water surface elevation has been measured in the field. The measured values will be compared with the values computed by the model to determine the model’s accuracy. In addition to being assigned a *Color* and a *Name*, each observation point is assigned the following data:

Location. The x, y real world location of the point needs to be specified. Observation arcs do not have these location attributes since several points define an arc.

Observed value. The observed value is the value that was measured in the field corresponding to the active measurement.

Confidence Interval. The confidence interval is the allowable error (\pm) between the computed value and the observed value. Model verification is achieved when the error is within the interval (\pm) of the observed value.

Confidence Level. The percent of confidence that the mean of the observed value will lie within confidence interval.

Angle. When a measurement for observation points is tied to a vector data set (as is the case with the *Velocity* measurement created in the previous section)

an angle needs to be specified. This angle is an azimuth angle with the top of the screen representing north when in plan view.

Table 17-1 Observation point values

x [ft]	y [ft]	Velocity [fps]	Interval [fps]	Confidence [%]
190	-369	3.5	0.25	95

One observation point should be created using the values in Table 17-1. In this case, the model will be verified if the computed value is ± 0.25 fps of the observed velocity, or between 3.25 and 3.75 fps. To create the observation point while still in the *Observation Coverage* dialog:

1. Type "Point 1" as the *Name* in the bottom line of the *Observation Points* spreadsheet. The *Observation Points* spreadsheet will always end with a blank line for the creation of additional points. Note, however, that there will be no blank line in the *Observation Arcs* spreadsheet since arcs cannot be created while in the *Observation Coverage* dialog.
2. Press *Enter* or *Tab* to create the new observation point.
3. Now that the observation point had been created, change the *X* coordinate to 190.0 and the *Y* coordinate to -369.0.
4. Enter 3.5 as the *Observed Val* and 0.25 as the *Conf. Int.* The *Conf. (%)* is already be set to 95. By default, after the *Observed Val* is entered, the *Obs* toggle for this point turns on. When the *Obs* toggle for a point or arc is on, it is said to be *Observed*.

An observation point has now been created at the location specified in the *Observation Coverage* dialog. However, no angle has been specified for this point. This angle can be specified in the *Observation Coverage* dialog or in the *Graphics Window*. To specify the angle in the *Graphics Window*:


1. Click *OK* to close the *Observation Coverage* dialog. A point with an arrow pointing up will appear in the *Graphics Window*. A calibration target is drawn next to the point.
2. Choose the *Select Feature Point*  tool from the *Toolbox*.
3. Rotate the point arrow approximately 120° by dragging the black dot at the end of the arrow clockwise. Do not worry if this angle is not exactly 120°. The arrow just needs to be pointing in the general direction the velocity meter was set up in the field. This is usually in the direction of flow. Figure 17-2 shows a close-up of *Point 1* with the arrow pointing up (0° angle) and then the position of the arrow at an angle of approximately 120°.



Figure 17-2 "Point 1" with an arrow angle of 0° and then rotated to 120°

17.6.1 Using The Calibration Target

A calibration target is drawn next to the observation point. The components of a calibration target are illustrated in Figure 17-3. These components are:

Target Middle. This is the target value that was measured in the field.

Target Extents. The top of the target represents the target value plus the interval while the bottom represents the target value minus the interval.

Color Bar. The color bar shows the error between the observed value and the computed value. If the bar is entirely within the target, the color bar is drawn in green. If the error is less than twice the interval, the bar is drawn in yellow. A larger error will be drawn in red.

For this example, the bar would be green if the computed value is between 3.25 and 3.75, yellow for values between 3.0-3.25 or 3.75-4.0, and red for values smaller than 3.0 or greater than 4.0.

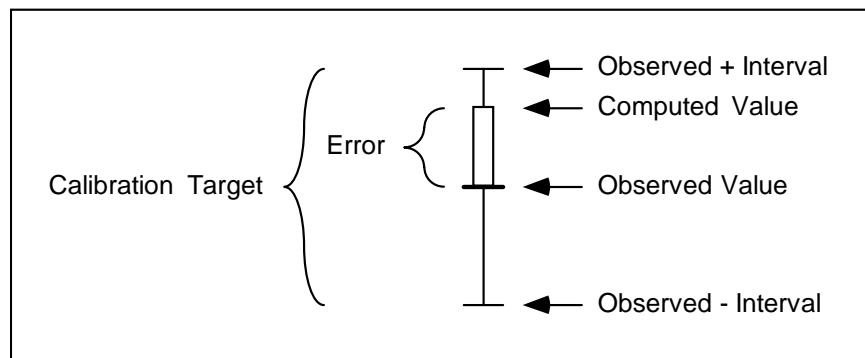



Figure 17-3 Calibration target

Now that the observation point has been created and a solution has been opened, the target appears. The color bar in this example is red with an arrow pointing down, indicating that the computed solution has a velocity below 3.0.

17.6.2 Multiple Measurements

Each observation point has attributes for all measurements. Similarly, each observation arc has attributes for each flux measurement. The highlighted measurement in the *Measurements* spreadsheet determines which attributes are shown in the *Observation Objects* spreadsheet.

For example, to create a new measurement:

1. Open the *Observation Coverage* dialog by choosing the *Select Feature Point*  tool from the *Toolbox* and double-clicking *Point 1*.
2. Type “WSE” as the *Name* in the bottom line of the *Measurements* spreadsheet. As with the *Observation Points* spreadsheet, the *Measurements* spreadsheet will always end with a blank line for the creation of additional measurements.
3. Press *Enter* or *Tab* to create the new measurement when finished typing to create the new measurement.
4. Select *water surface* as the *Data Set*.

Note that this new measurement is now the *Active* measurement and it is also highlighted. Several measurements can exist at a time; however, calibration targets will only be displayed in the *Graphics Window* for *Observed* points in the *Active* measurement.

Now look at the *Observation Points* spreadsheet. The *Name*, *Color*, and *X* and *Y* coordinates have remained the same for *Point 1*, however, the *Observed Val* and *Conf. Int.* have been reset to their default values. There is no *Angle* column as well since this new measurement is tied to a scalar data set. These attributes are for the measurement named *WSE*. To view the observation point attributes previously specified for the *Velocity* measurement, simply click the *Velocity* measurement to highlight it in the *Measurements* spreadsheet.

Do not delete the *WSE* measurement since both it and the *Velocity* measurement will be used to calibrate the model. Before continuing, make the *Velocity* measurement the *Active* measurement.

17.7 Reading a Set of Observation Points

Using the steps defined above, multiple observation points can be created. However, this process could become tedious for a large set of points. Normally, the data defining the points will be in spreadsheet format and can simply be copied and pasted in the *Observation Points* spreadsheet. To do this:

1. Open the file *observepts.obt* in a spreadsheet program.
2. Highlight and copy the data from the column labeled “Name” to the first column labeled “int” for *Point 2* to *Point 8*. The data for *Point 1* does not need to be copied since *Point 1* has already been created.
3. Return to *SMS* and make sure the *Velocity* measurement is selected.
4. Select the *Name* of the bottom row of the *Observation Points* spreadsheet as the starting cell for the data to be pasted and paste the copied data into the *Observation Points* spreadsheet.
5. Click *OK* to close the *Observation Coverage* dialog.

Seven new observation points appear in the *Graphics Window*. The new points are distributed around the finite element mesh, as shown in Figure 17-4.

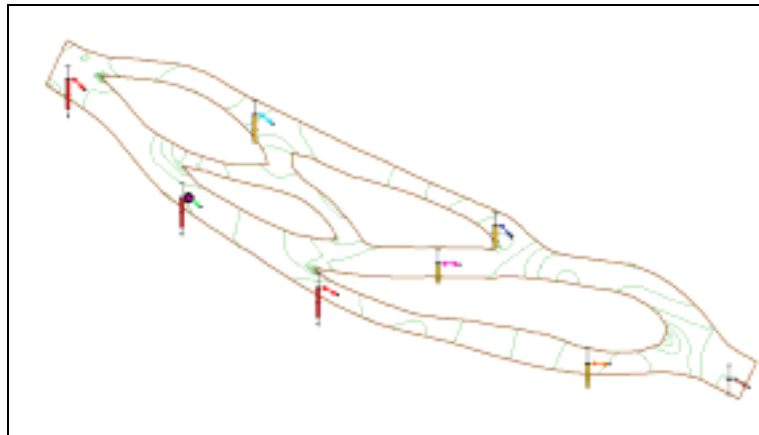


Figure 17-4 The observation points created from the file *observepts.obt*

Now, the observed values and the confidence interval for the *WSE* measurement need to be specified. To do this:

1. Open the *Observation Coverage* dialog by double-clicking on one of the points.
2. Using the same spreadsheet file opened earlier, *observepts.obt*, highlight and copy the data from the column labeled “wse” to the second column labeled “int” for *Point 1* to *Point 8*.


3. Return to *SMS* and make sure the *WSE* measurement is selected.
4. Select the *Observed Val* of the top row of the *Observation Points* spreadsheet as the starting cell for the data to be pasted and paste the copied data into the *Observation Points* spreadsheet.

To view that calibration targets for the *WSE* measurement, make the *WSE* measurement the *Active* measurement and close the *Observation Coverage* dialog by clicking the *OK* button. The points that appear in the *Graphics Window* do not have arrows since the active measurement is observing a scalar data set.

When calibrating a model the goal is to calibrate the model so that the computed values from the model fall within the confidence intervals of the observed field data for all measurements. At times this is difficult and personal discretion is required to determine when the model has sufficiently been calibrated. Before continuing, make the *Velocity* measurement the *Active* measurement.

17.8 Generating Error Plots

SMS can create several types of plots to analyze the error between the computed and observed values. To create a *Computed vs. Observed Data* plot and an *Error Summary* plot:

1. Select *Display / Plot Wizard...* .
2. Choose *Computed vs. Observed Data* as the *Plot Type*.
3. Click *Next* and choose *Velocity* as the measurement.
4. Click *Finish* to close the *Plot Wizard* and generate the plot.

Create another plot of the *Velocity* measurement, but this time choose *Error Summary* as the *Plot Type*.

Both plots have now been created. Each plot exists in a separate window that can be resized, moved, and closed at any time. The plots that appear are shown in Figure 17-5.

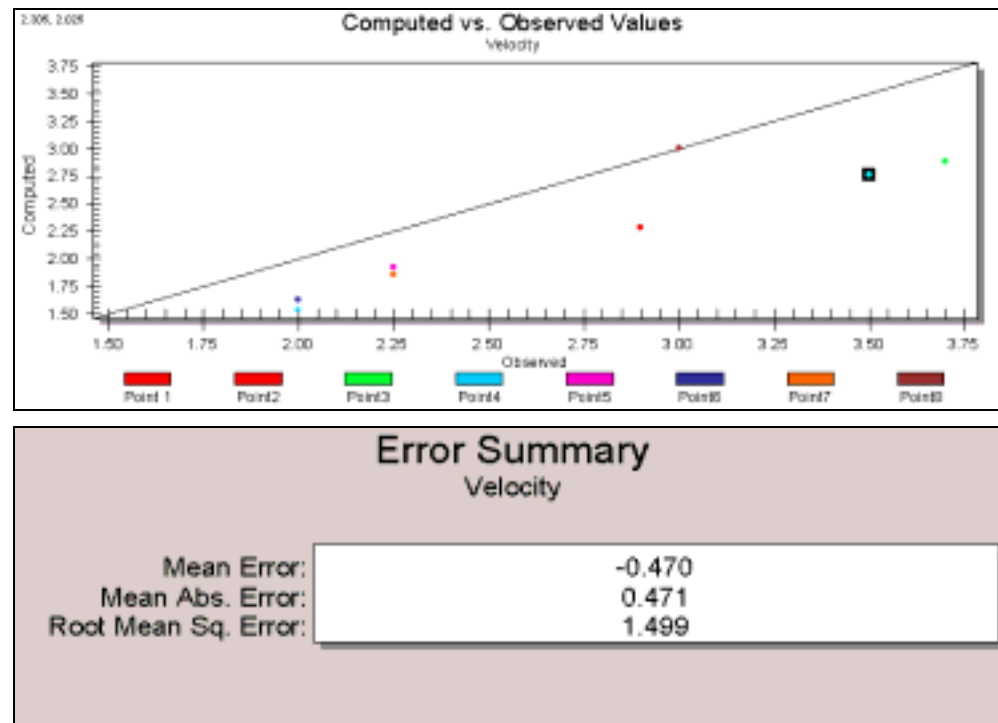


Figure 17-5 Computed vs. Observed Data and Error Summary plots

17.8.1 Plot Data

More plots can also be created for the *WSE* measurement or the current plots can be edited. To edit a plot:

1. Right-click the *Error Summary* plot and select *Plot Data...* from the menu.
2. Select *WSE* as the measurement.
3. Click *OK* to close the *Data Options* dialog.

The *Error Summary* plot is now updated using the data from the *WSE* measurement.

17.8.2 Using The Computed vs. Observed Data Plot

In the *Computed vs. Observed Data* plot, a symbol is drawn for each of the observation points. A point that plots on or near the diagonal line indicates a low error. Points far from the diagonal have a larger error. The position of the points relative to the line gives an indication whether the computed values are consistently higher or lower than the observed values. In this case, all points are below the line indicating that all computed velocities are lower than observed values.

Change the measurement for the *Computed vs. Observed Data* plot to the *WSE* measurement by following the steps above defined in section 17.8.1. Now, all points plot above the line indicating that all computed water surface elevations are above the observed values.

17.8.3 Using The Error Summary Plot

In the *Error Summary* plot, the following three types of error norms are reported:

Mean Error. This is the average error for the points. This value can be misleading since positive and negative errors can cancel.

Mean Absolute Error. This is the mean of the absolute values of the errors. It is a true mean, not allowing positive and negative errors to cancel.


Root Mean Square. This takes the sum of the square of the errors and then takes its square root. This norm tends to give more weight to cases where a few extreme error values exist.

17.9 Calibrating The Model

The values in this solution for both measurements are not within the calibration targets. To achieve better calibration, the material properties will be changed and then the model will be re-run. Since the errors through the main channel for the *Velocity* measurement are negative, indicating that the observed velocities are larger than those computed by the model, we want to change the parameters in such a way as to increase the velocity in these locations (eddy viscosity and/or Manning's n). Increasing the velocity at these locations should also decrease the water surface elevation.

17.9.1 Editing The Material Properties

Decreasing the eddy viscosity values can increase these computed velocities. To decrease the eddy viscosity:

1. Switch to the *Mesh*  module.
2. Select *FESWMS / Material Properties...*
3. Change the Vo (kinematic eddy viscosity value) from 10.0 to 1.5.
4. Click *Close* to close the *FESWMS Material Properties* dialog.

17.9.2 Computing a New Solution


To compute the new solution:

1. Go to *File / Save As...* and save the project as *observe2.spr*.
2. Run flo2dh on the new simulation.

If you are using *SMS* in *Demo* mode, you will not be able to save the simulation. However, this second simulation has been saved in the *output* directory. You can open the second simulation if you need to.

17.9.3 Reading The New Solution


The new *FESWMS* solution should now be opened and the errors associated with it should be plotted. To open the second solution:

1. In the *Mesh*  module select *Data / Data Browser...*
2. Click the *Import...* button.
3. Select the file named *observe2.flo* and click *OK*. If you are running *SMS* in *Demo* mode, this solution file is in the *output* directory for this tutorial.
4. A new solution set is created for this second simulation. It is named *observe2.flo* after the file name and contains the same data sets as the previous solution.
5. Click *Done* to close the *Data Browser*.

The plots will automatically update to show the errors for to the solution that was just opened.

17.9.4 Fine-tuning the model

The verification targets now show that six points for the *Velocity* measurement are within the allowable range and two points are above the range, but still in the yellow range. There are no points more than the two times the variation above the observed value (red targets). Looking at targets for the *WSE* measurement four points are within the allowable range and four are below the range with one point being more than two times the variation below the observed value. Since the values for the *Velocity* measurement that are unacceptable are now higher than the observed values and the values for the *WSE* measurement that are unacceptable are now lower than the observed values, the correction made was too drastic. Specifically, the eddy viscosity was lowered too much and it needs to be raised. To compute another solution:

1. In the *Mesh*  module select *FESWMS / Material Properties...*
2. Change the *Vo* (kinematic eddy viscosity value) from 1.5 to 6.0.
3. Click Close to close the *FESWMS Material Properties* dialog.
4. Save and run a third simulation of flo2dh.
5. Open the third simulation. If you are running *SMS* in *Demo* mode, this third simulation can be read from the *output (observe3.flo)* folder.

After this third simulation is opened, all the observation point targets for the *Velocity* measurement should be within the acceptable intervals.


Now make the *WSE* measurement the *Active* measurement. All but one point is within the acceptable interval. To get this one point with range, compute another solution:

1. Change the kinematic eddy viscosity value (*Vo*) to 7.0.
2. Save and run a fourth simulation.
3. Open the fourth simulation. If you are running *SMS* in *Demo* mode, this fourth simulation can be read from the *output (observe4.flo)* folder.

The calibration process is now complete. It will not always be possible to get all observation points for each measurement to be within the acceptable confidence interval. Therefore, it will have to be decided which measurements and which points are the most important to have within the acceptable range.

17.10 Using The Error Vs. Simulation Plot

When performing trial-and-error verification, it is often important to keep track of the error trend as new solutions are repeatedly computed. *SMS* provides a special verification plot to simplify this task. To create this plot:

1. Select *Display / Plot Wizard...* .
2. Choose *Error vs. Simulation* as the *Plot Type* and click *Next*.
3. Select *Velocity* as the measurement.
4. *SMS* will create a plot with one point for each simulation. The order of the points in the plot will follow the order solution sets in the *Solutions* list box. The solution at the top will be first. Use the *Move Up* and *Move Down*

buttons to change this order. The default order is the order that they were read in.

5. Click *Finish* to close the *Plot Wizard* and generate the plot.

A new plot appears showing the *Error vs. Simulation*, as shown in Figure 17-6. Notice for the *Velocity* measurement that the errors decrease as each simulation was performed until the final solution where the errors slightly increase. This slight increase in error with the *Velocity* measurement was required to get that last observation point for the *WSE* measurement within the acceptable range. Generally, if the errors increase, then the model is not improving.

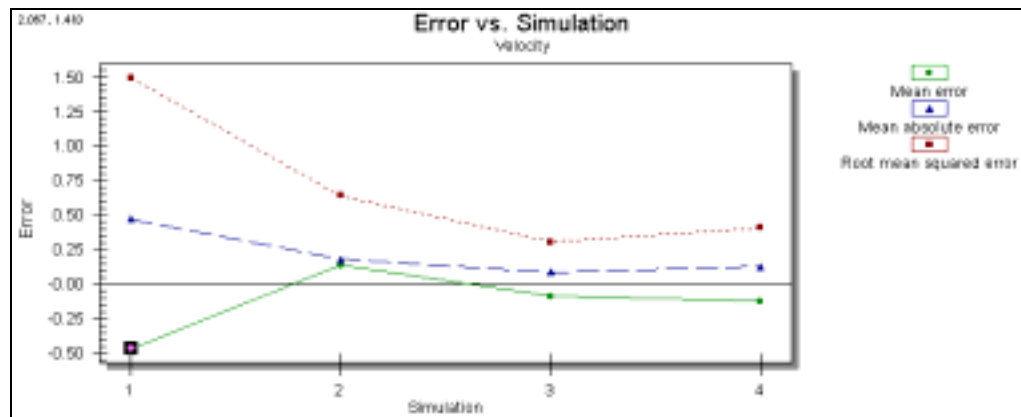


Figure 17-6 Error vs. Simulation plot for Velocity

Create another *Error vs. Simulation* plot using the *WSE* measurement. The errors for this measurement changed dramatically from solution to solution since parameters were first changed to calibrate points for the *Velocity* measurement. However, the general trend was a decrease in error. This plot is shown in Figure 17-7.

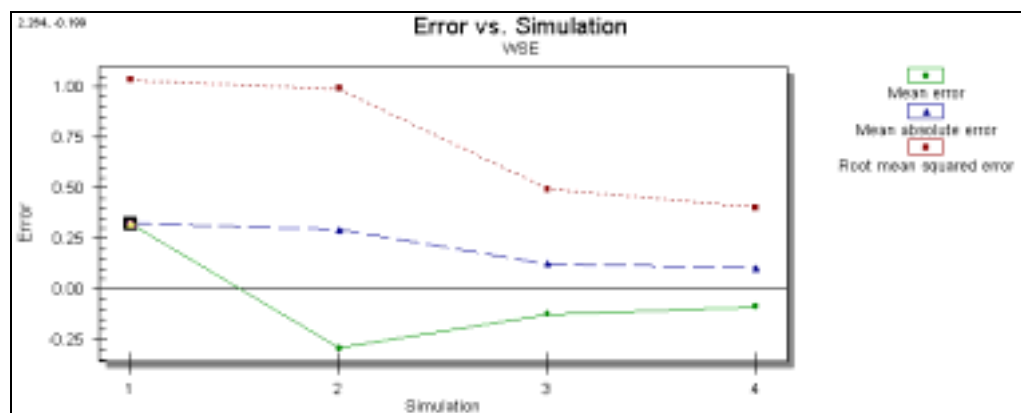





Figure 17-7 Error vs. Simulation plot for WSE

17.11 Generating Observation Profile Plots


Observation profile plots are used to view data set values along observation arcs. The first observation arc to be created will be used to create a profile of the main channel. To create this arc:

1. In the *Map*  module create a new observation coverage named “Profiles.” When an observation arc is being created, observation points may be clicked joining them to the arc. Creating observation arcs on a separate coverage will keep this from happening. (Observation points and arcs can exist on the same coverage.)
2. Choose the *Create Feature Arc*  tool from the *Toolbox*.
3. Create an arc down the main channel, as shown in Figure 17-8. Remember to double-click the last point to end the arc.

When the plots are drawn, they will use the name and color associated with the observation arc. To change the name and color of the arc:

1. Choose the *Select Feature Arcs*  tool from the *Toolbox*.
2. Double-click on the profile arc.
3. In the *Observation Coverage* dialog, change the *Name* of the arc to “river profile” in the *Observation Arcs* spreadsheet and turn off the *Obs* toggle since the arc is being used to create a profile plot and not to observe a flux (leave its *Color* red).
4. Click *OK* to close the *Observation Coverage* dialog.

Three more arcs need to be created, each across a section of the river. These arcs will be used to create cross section plots. To create these arcs:

1. Select the *Create Feature Arcs*  tool from the *Toolbox*.
2. Create each of the cross section arcs, as shown in Figure 17-8. Note: When creating these, DO NOT click ON the profile arc, as this would split it.

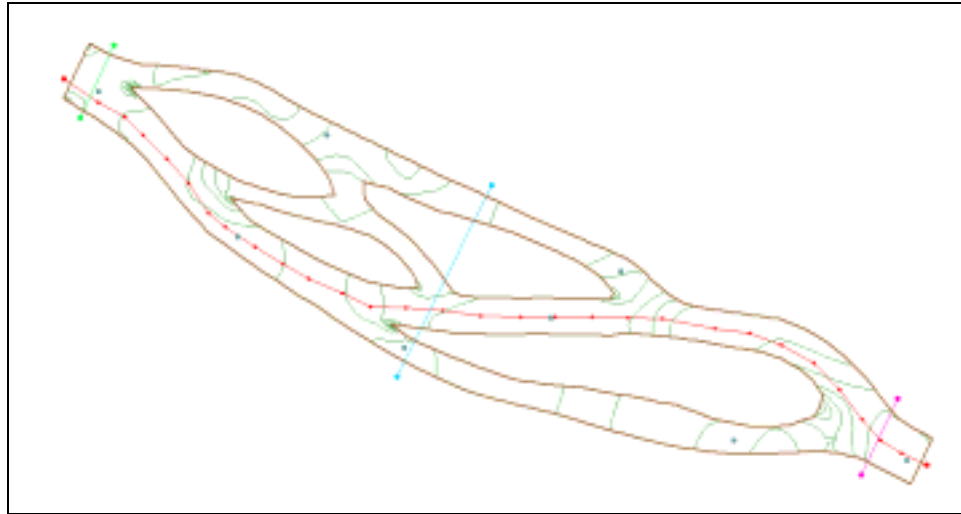



Figure 17-8 Profile and cross-section arcs

With the cross sections created, open the *Observation Coverage* dialog and assign a unique color and an appropriate name to each arc.

With the arcs created, the plots can now be generated. To do this:

1. Select *Display / Plot Wizard...* .
2. Choose *Observation Profile* as the *Plot Type* and click *Next*.
3. Turn on the *Use selected data sets* option, and check only the *elevation* data set in the *Generic Solution* and the *water surface* data set in the *observe4.flo* solution.
4. Turn off the three cross-section arcs in the *Arcs* spreadsheet by turning off their corresponding *Show* toggles.
5. Click *Finish* to close the *Plot Wizard* and generate the plot.

The profile plot of the geometry of the stream should appear as shown in Figure 17-9.

To view the velocity distribution across the three cross sections create a new *Observation Profile* plot. Turn on the *velocity mag* data set in the *observe4.flo* solution, and *Show* only the three cross section arcs.

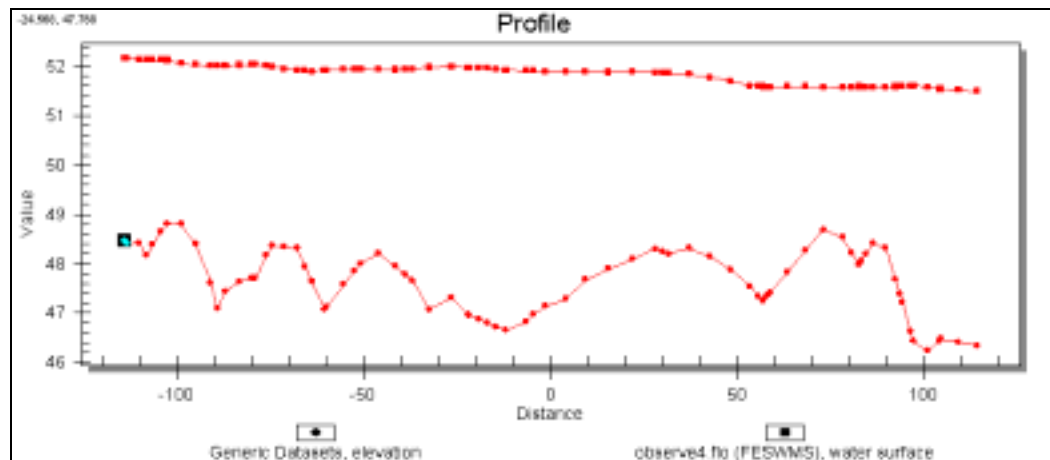


Figure 17-9 Observation Profile Plot

17.12 Generating Time Series Plots

As mentioned earlier, observation arcs are used to compute fluxes. One flux value that is often observed and measured in the field is flow rate. Observed flow rates can be used in model calibration in the same way observed velocities and water surface elevations are used. In addition to normal model calibration, *Time Series* plots can be generated showing how the flow rate flux changes with time. This type of time series plot is commonly known as a hydrograph. Hydrographs created using calculated data from the model are useful to see if the model properly predicts flow rate patterns. To create a *Time Series* plot:

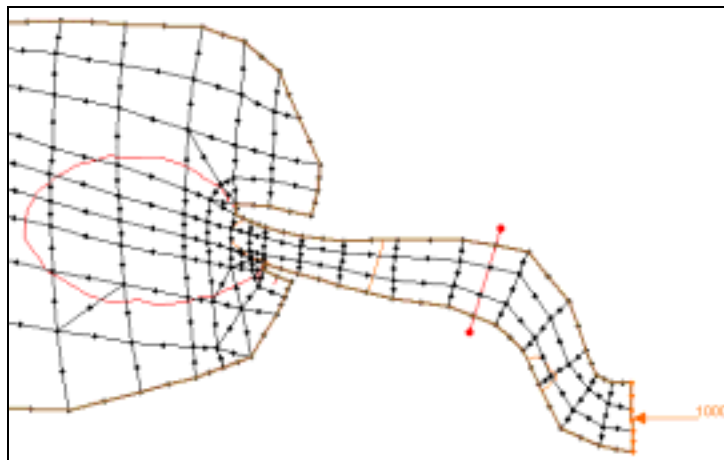


Figure 17-10 Observation arc across noyo1.spr mesh

1. Open the file *noyo1.spr* from the *tut10* directory. If geometry data is still open, you will be warned that the existing mesh will be deleted. If this happens, click *OK* to the prompt.

2. Create a new observation coverage called “Fluxes.”
3. Create an observation arc across the mesh as shown in Figure 17-10.
4. Open the *Observation Coverage* dialog and change the name of the default flux measurement to “Flow Rate” and select *water depth* as the scalar data set. Leave *velocity* as the vector data set.
5. Click *OK* to close the *Observation Coverage* dialog.
6. Open the *Plot Wizard*.
7. Choose *Time Series* as the *Plot Type* and click *Next*.
8. Click *Finish* to close the *Plot Wizard* and generate the plot.

A new window opens with the *Time Series* plot of the *Flow Rate* flux measurement. This plot should appear similar to the one in Figure 17-11.

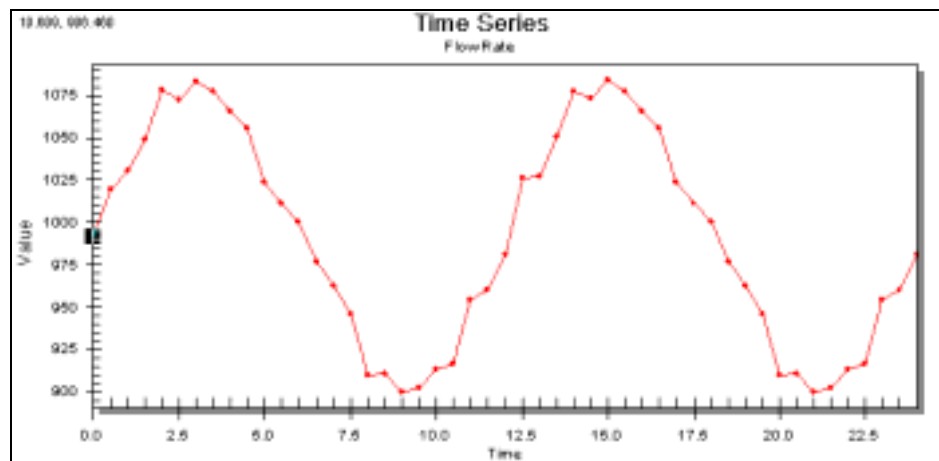


Figure 17-11 Time Series plot of Flow Rate.

17.13 Conclusion

This concludes the *Observation Coverage* tutorial. You may continue to experiment with the program or you can exit *SMS*. To quit *SMS* at this point:

- Select *File / Exit*. If prompted to confirm, click the *Yes* button.

Sensitivity Analysis

18.1 Introduction

This lesson analyzes the effects of changes in Manning's roughness coefficients and of kinematic eddy viscosity on various channel arrangements. Understanding the effects of Manning's roughness and eddy viscosity are useful in model calibration.

Either *RMA2* or *FESWMS* may be used for this lesson.

18.2 Simple Channel With Single Material

A flume 800 meters by 100 meters has been prepared for use in this lesson. The flowrate is set at 800 m³/s. The downstream water surface elevation is 1 m. This flume has no slope and is comprised of a single material.

18.2.1 Open The Simulation

To open the file that contains the necessary mesh.

1. Select *File / Open*.
1. Use the *tut18\rma2* directory if you are using *RMA2* and the *tut18\feswms* directory for *FESWMS*. Open *flumea1.spr*. If geometry data is still open

from a previous tutorial, you will be warned that the existing mesh will be deleted. If this happens, click *OK* to the prompt.

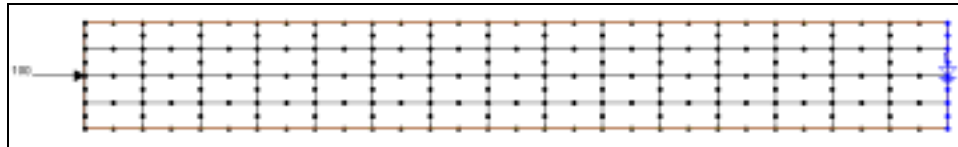



Figure 18-1 The mesh flumea1.

18.2.2 Running The Model

The correct material properties have been set for the initial run. You will need to run the model with the current settings. For instructions on running RMA2, see the Basic RMA2 Analysis. For instructions on how to run FESWMS, see the Basic FESWMS Analysis.

18.2.3 Importing Solutions


Solutions are imported into SMS for visualization, and/or calibration. To import a solution into SMS:

1. Make sure that you are in the *Mesh*  Module.
2. Select *Data / Data Browser*.
3. Click on the *Import* Button.
4. If using *RMA2* select the file *flumea1.sol*. If using *FESWMS* select the file *flumea1.flo*.
5. Click the *OK* button, and then *Done* to exit the *Data Browser*.

18.2.4 Creating Profile Plots

Before making a profile plot, it is necessary to create an observation coverage. In addition, an observation arc must be created to define the profile to plot. To create an observation coverage and profile arc:

1. Go to the *Map*  module.

2. Select *Feature Objects / Coverages*.
3. Change the *Coverage type* to *Observation*.
4. Click the *OK* Button.
5. Select the *Create Feature Arc*  tool from the toolbox.
6. Create an arc down the center of the flume as shown in Figure 18-2

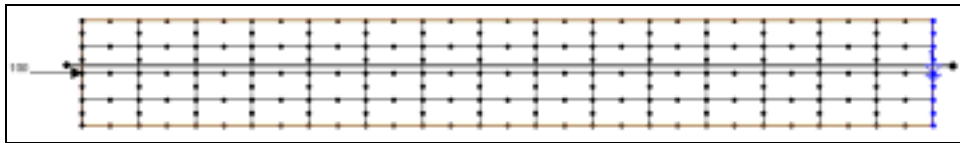



Figure 18-2 Mesh with Observation Arc.

In *SMS*, profile plots can be created to visualize the results of a model run. To create a profile plot:

1. Select *Display / Plot Wizard*.
2. Make sure the plot type is *Observation Profile*.
3. Click on the *Next* Button.
4. Choose *Use Selected Dataset*.
5. Turn on the *water depth* function for the current solution. Functions are turned on and off by clicking on them. Make sure the other functions are turned off.
6. Click the *Finish* Button to exit the *Plot Wizard* dialog.

18.2.5 Varying Manning's Roughness

The next step is to change the material properties and rerun the model in order to compare the results. To change the material properties:

1. Switch to the *Mesh*  module
2. If using *RMA2*, select *RMA2 / Material Properties*.
If using *FESWMS*, select *FESWMS / Material Properties*.
3. Change the *material_01* Manning's *n* (*n1* and *n2* for *FESWMS*) value to 0.045.

4. Click the *Close* button to close the *Materials Properties* dialog.
5. Select *File / Save As*.
6. Save the new project as *flumea2.spr*.
7. Repeat sections 18.2.2 and 18.2.3 to rerun the model with the new roughness value.

Repeat steps 2-7 except change the n value to 0.065, and save the file as *flumea3.spr*.

18.2.6 Updating The Plot

Currently, the plot only displays the first solution. To add the new solutions:

1. Right click on the Profile plot made earlier and select *Plot data*.
2. Turn on the *Water Depth* functions for each solution. Make sure all of the other functions are turned off.
3. Click the *OK* Button.

The plot should now look like Figure 18-3. The plot demonstrates the fact that as the roughness increases, the upstream water surface elevation increases.

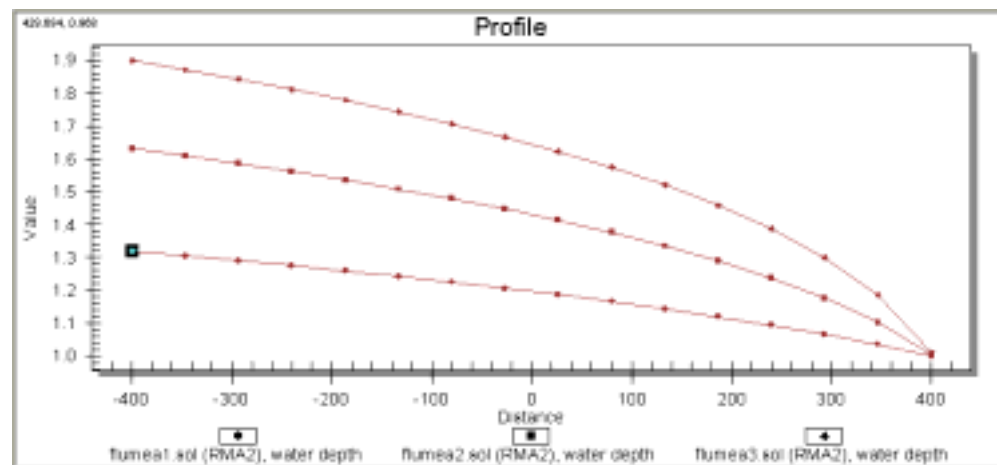


Figure 18-3 Water depth with varied roughness parameters for.

18.2.7 Changes in Eddy Viscosity

Eddy Viscosity is another parameter that can be modified to alter the model's solution. This section will analyze the effects of various eddy viscosities while keeping Manning's coefficient constant. To setup the first run:

1. First, delete the old solutions with the *Data Browser*.
2. If using *RMA2*, select *RMA2 | Material Properties*.
If using *FESWMS*, select *FESWMS | Material Properties*.
3. Change the *material_01* Manning's *n* value to 0.035.
4. If using *RMA2*, change the *Eddy Viscosity* (E) to 5.0.
If using *FESWMS*, change the *Viscosity* (Vo) to 1.0 m²/s.
5. Click the *Close* button.
6. Select *File | Save As*.
7. Save the project as *flumeb1.spr*.
8. Repeat sections 18.2.2 and 18.2.3 to rerun the model with the new model parameters.

Now create two new solutions using steps 1-7. For *FESWMS* use viscosities of 10 and 100 m²/s. For *RMA2* use viscosities of 100 and 500,000. Name the files as *flumeb2.spr* and *flumeb3.spr*. Import the new solutions into the *Data Browser* and turn off every thing but the water depth functions in *Plot Data* dialog (right click on plot).

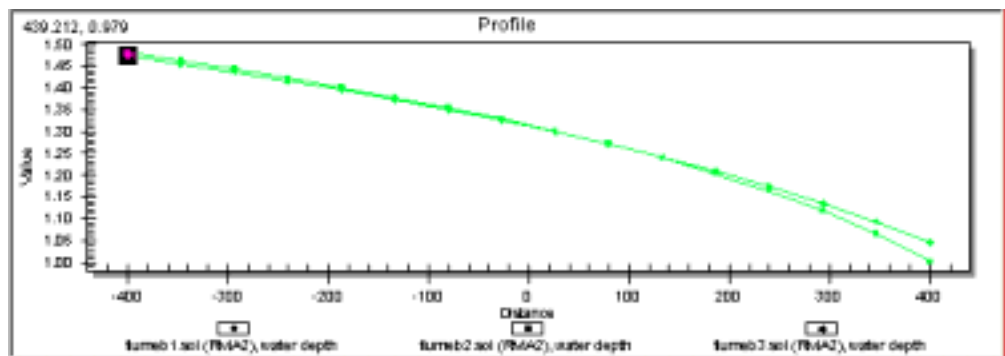


Figure 18-4 Various Eddy Viscosities with $n = 0.035$ using *RMA2*.

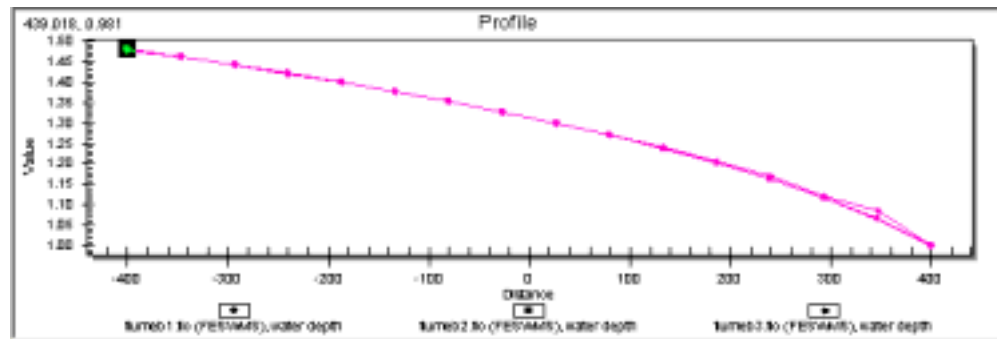


Figure 18-5 Eddy Viscosities of 1, 10, 100 m²/s using FESWMS.

The results in *RMA2* changed little even for the unrealistic value of 500,000. *FESWMS* had no discernable difference even for values as high as 100 m²/s.

18.3 Constrained Flume With Single Material

The second channel was designed to show the effect of roughness coefficients and eddy viscosities when large velocity gradients occur in the longitudinal flow direction. This channel has the same dimensions as our first flume, but it is constricted to 20 m wide through the middle. The channel has gradual contractions and expansions above and below the constricted section. The flowrate will remain 800 m³/s. The downstream water surface elevation will remain 1 m.

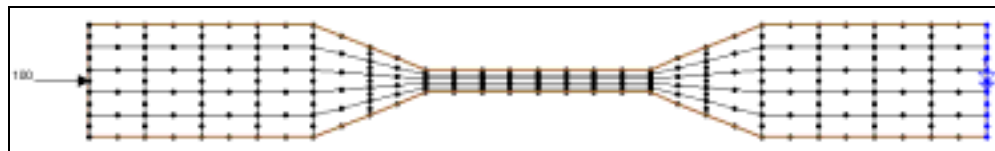


Figure 18-6 Test Channel #2

18.3.1 Open The Simulation

To open the new mesh:

1. Select *File / Open*.
2. Select the file *flumec1.spr*.

18.3.2 Varying Manning's Coefficient

Repeat the same procedure as was outlined in sections 18.2.2 to 18.2.5. First, run the model as configured. Use *n* values of 0.045 and 0.065 for the subsequent model runs. Save the files as *flumec2.spr* and *flumec3.spr*. Make sure to add all three solutions into the profile plot. When finished, the plot should look like Figure 18-7.

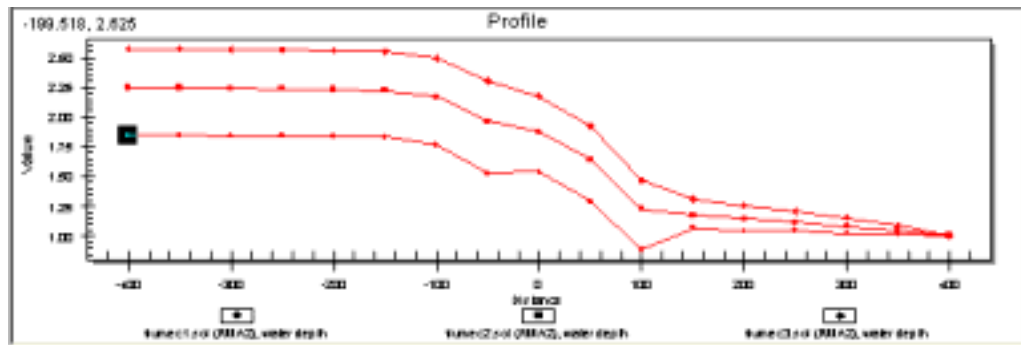


Figure 18-7 Constricted flume with various roughness factors.

18.3.3 Varying Eddy Viscosities

To analyze the effect of changing eddy viscosities:

1. In the *Material Properties* dialog change Manning's n value to 0.035.
2. If using *RMA2*, change the *Eddy Viscosity* (E) to 5.

If using *FESWMS*, change the *Viscosity* (Vo) to 1 m²/s.

3. Select *File / Save As*.
4. Save the file as *flumed1.spr*.
5. Repeat sections 18.2.2 and 18.2.3 to rerun the model with the new model parameters.

Now create two new solutions using steps outlined above. For *FESWMS* use viscosities of 10 and 100 m²/s. For *RMA2* use viscosities of 100 and 500. Name the files *flumed2.spr* and *flumed3.spr* respectively. Import the new solutions into the *Data Browser* and turn on the water depth functions in *Plot Data* dialog.

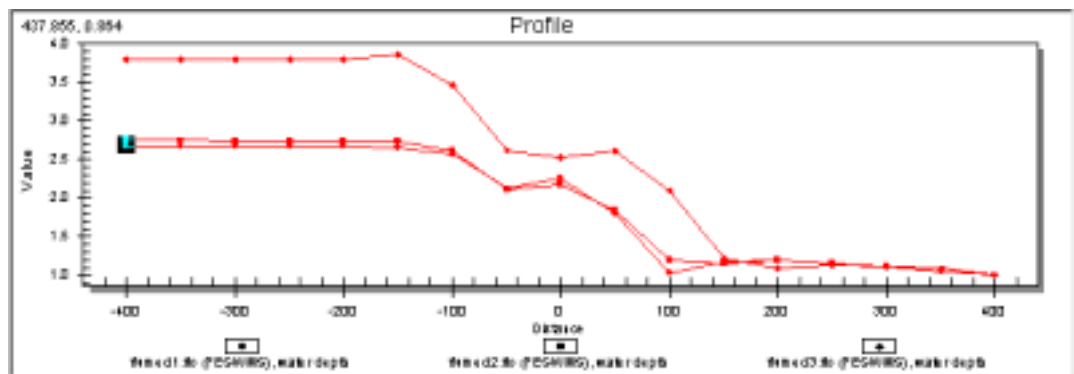


Figure 18-8 FESWMS run of the constricted flume at various eddy viscosities

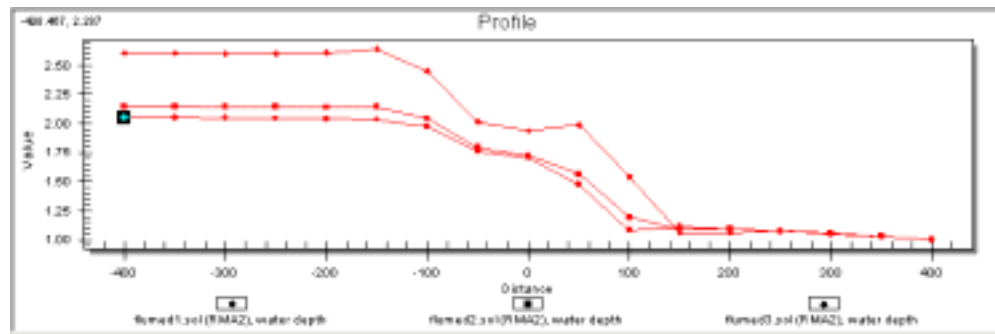


Figure 18-9 RMA2 run of the constricted flume at various eddy viscosities.

As shown in Figure 18-8 and Figure 18-9, eddy viscosities have a much larger effect when there are large longitudinal velocity gradients. For realistic values of eddy viscosity, differences in depth at the upstream end of the channel are small.

18.4 Simple Channel With Two Materials

This channel has the same dimensions and boundary conditions as the first one. The elements are smaller and the channel has two material types rather than one. We will examine the effects the lateral roughness variation has on velocity.

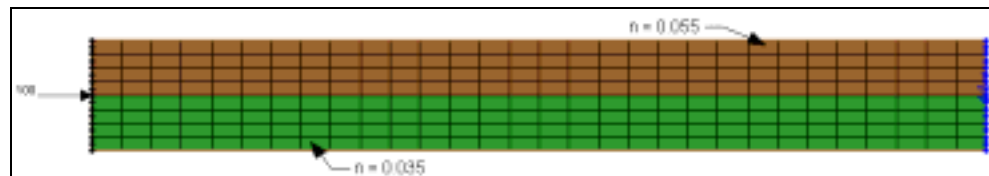


Figure 18-10 Channel #3 simple flume with two materials

This time specific instructions will not be given. If you don't remember how to do something, look back at the lesson for help.

1. Open the file *flumee1.spr*.
2. Run the simulation with the current settings.
3. For *RMA2* rerun the model with viscosities of 500, 5000, and 50,000 for both materials.

For *FESWMS* rerun the model with viscosities of 5, 50, and 100 for both materials.

Name the simulation files *flumee2.spr*, *flumee3.spr*, and *flumee4.spr*.

4. Create an observation coverage.

5. Create an observation arc across the flume at about 200 m from the downstream boundary as shown in Figure 18-11.
6. Create an observation profile plot turning on the velocity mag function for each solution.

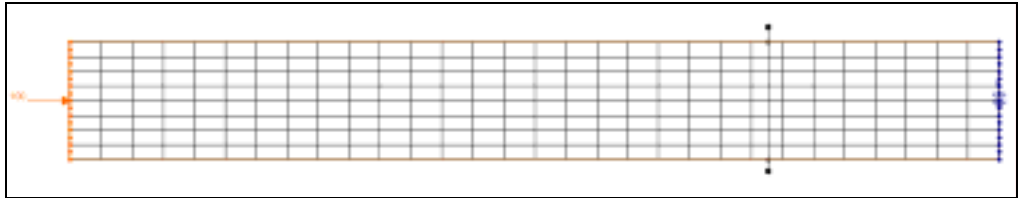


Figure 18-11 Channel #3 showing placement of observation arc.

The plot should look like Figure 18-12 or Figure 18-13. As you can see in the graph, smaller eddy viscosities allow larger transverse velocity gradients to appear in the solution.

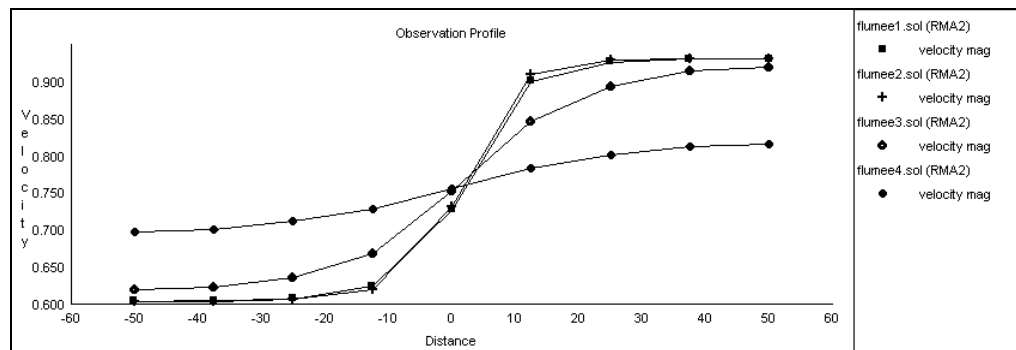


Figure 18-12 Profile plot of RMA2 solution for various eddy viscosities for channel #3.

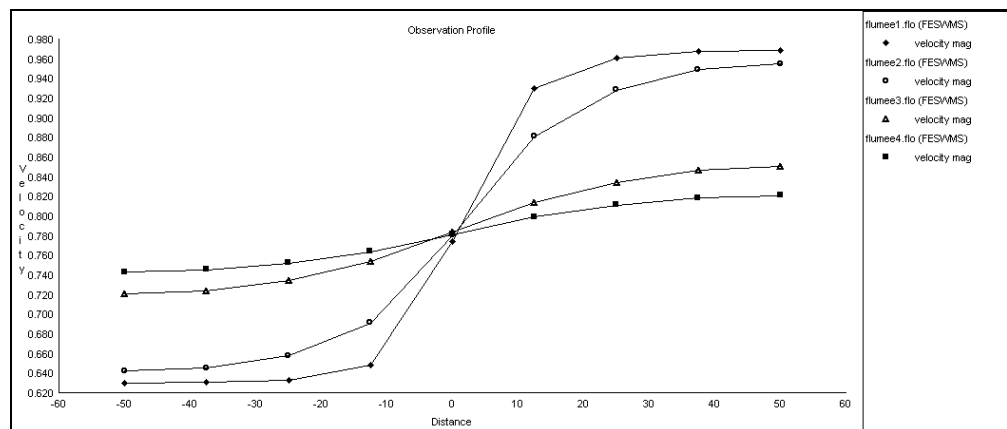


Figure 18-13 Profile plot of FESWMS solution for various eddy viscosities for channel #3.

18.5 Conclusion

This concludes the *Sensitivity Analysis* tutorial. You may wish to experiment further with different channel arrangements and watch the effects of changing roughness and viscosity values. This concludes the tutorial. If you wish to exit *SMS* at this point:

- Choose *File / Exit*.